POLITECNICO DI TORINO

FEM SIMULATION OF FRICTION STIR WELDING PROCESS ON SYSWELD

PROF. DE MADDIS MANUELA
PROF. RUSSO SPENA PASQUALE
PROF. LOMBARDI FRANCO

WANG JINGANG

Academic Year 2020 - 2021

ACKNOWLEDGMENTS

I would like to express my deepest gratitude to my supervisor, Professor Manuela De Maddis. She made instructive comments and suggestions on the research direction of my thesis. During the process of writing the thesis, she gave timely and careful guidance on the difficulties and doubts I encountered, put forward many useful suggestions for improvement. I wish to thank all my colleagues, especially Mr.Panza Luigi, whose assistance was a milestone in the completion of this project.

I wish to show my gratitude to Professor Pasquale Russo Spena and his team to carry out the friction stir welding expeirments. Without their tireless help, the project's goals could not have been achieved.

Thanks to Professor Franco Lombardi for accepting me in the research group and introducing such a professional team to work with me.

Last but not least, thanks my families to support me and encourage me to spend this difficult period.

ABSTRACT

Friction Stir Welding is a solid-state welding technology that uses the thermo-mechanical effect of a rotary tool on the base material to achieve overall welding.

It has been widely used in the welding applications of the low melting point alloy such as aluminium alloy, but for the high melting point and high strength alloys, it is still in the research stage.

With implementation of Friction Stir Welding experiment in laboratory, the aim of this work is to create a virtual simulation of this cutting-edge welding technology. An appropriate process modelling will lead to a good understanding of material flow and temperature field distribution, moreover the prediction of Friction Stir Welding process under different condition will be available. However, it is difficult to numerically simulate this technology because the process is highly complex. Indeed, this is a fully coupled thermomechanical problem with large plastic deformation, heat flow, and non-linearity contact.

In this study, the contact and the friction were predicted using a finite element simulation of the coupled Eulerian Lagrangian formula. Furthermore, within the available range of FEM computing software, the main differences between Abaqus, COMSOL and Sysweld have been discussed. After comparing the calculation methods, complexity of the models, approximation methods adopted and contextualization of the final results, Sysweld software has been chosen to build the numerical model.

The simulation procedure is articulated as model building, model mesh, finite element group definition, process parameter definition, material parameter input, simulation execution and simulation result examination. With the development of simulation procedures, a further research to check the alignment of simulation result and experimental measurements has been carried out.

This numerical model enables the possibility to virtually explore different materials and process conditions, thus speeding up the design phase of advanced joints that require an innovative joining technique such as the Friction Stir Welding process.

TABLE OF CONTENTS

Page

LIST OF FIGURES

Figure Page

1 CHAPTER

SOLID-STATE WELDING PROCESS

1.1 Introduction

Solid-state welding, SSW, is a process which the welding temperature is lower than the melting temperature of the base metal and the filler metal, and the pressure is applied for the mutual diffusion of atoms.[1] The solid-state welding method can avoid the occurrence of some phase transformation and reduce the formation of some compounds at the welding interface, so as to enhance the interface bonding strength to the greatest extent.

The solid-state welding can be classified by the types of reaction which will happen during the welding process: Electrical, Chemical or Mechanical.

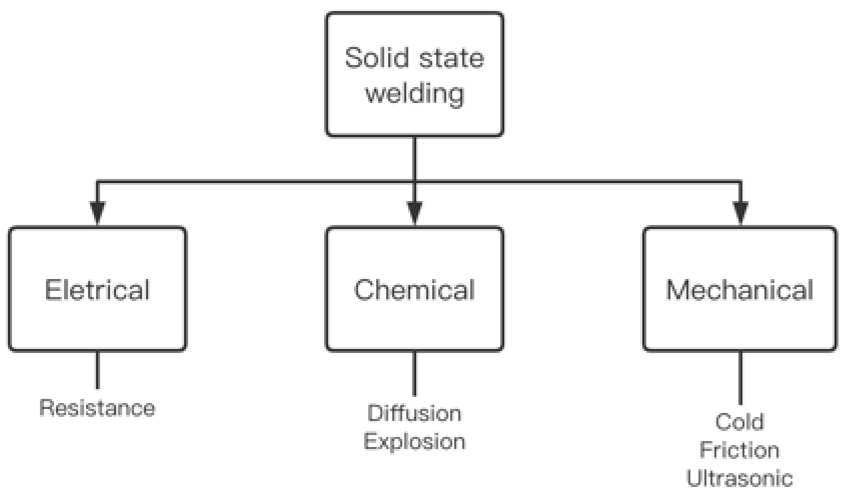


Figure 1.1.1 Classification

1.2 Resistance welding

Resistance welding is a method of mounting the workpieces to be welded between two electrodes, applying electric current, and heating it to a molten or plastic state by using the resistance thermal effect generated by the current flowing through the contact surface and adjacent area of the workpiece, so as to form metal bonding. [2]

Advantages:

1. When the nugget is formed, it is always surrounded by the plastic ring, and the molten metal is isolated from the air, so the metallurgical process is simple.
2. The heating time is short and the heat is concentrated, so the heat affected zone is small and the deformation and stress are small. Generally, it is not necessary to arrange the correction and heat treatment process after welding.

3. It does not need filler metal such as welding wire and electrode, and welding materials such as oxygen, acetylene and hydrogen, so the welding cost is low.
4. High productivity, no noise and harmful gas, in mass production, can be woven together with other manufacturing processes to the assembly line. However, flash butt welding needs to be isolated due to spark splashing.
5. Simple operation, easy to realize mechanization and automation, improve the working conditions.

Disadvantages:

1. At present, there is still a lack of reliable nondestructive testing methods, and the welding quality can only be checked by the destructive test of process samples and workpieces, as well as by various monitoring technologies.
2. The lap joint of spot and seam welding not only increases the weight of the component, but also forms an angle around the nugget of two plates, resulting in low tensile strength and fatigue strength of the joint.
3. The high power requirement, mechanization and automation of the equipment make the equipment more expensive and difficult to maintain, and the common high-power single-phase AC welding machine is not conducive to the balanced operation of the power grid.

There are four main resistance welding methods, namely spot welding, seam welding, projection welding and butt welding.

1.2.1 Spot welding

Spot welding refers to the welding method of using cylindrical electrode to form a spot between the contact surfaces of two overlapped workpieces. During spot welding process, the workpiece is pressed to make close contact, and then the current is switched on. Under the action of resistance heat, the contact part of the workpiece is melted, and the solder joint is formed after cooling.

Spot welding is mainly used for sheet metal stamping parts with thickness less than 4mm, especially for car body, carriage and aircraft fuselage.

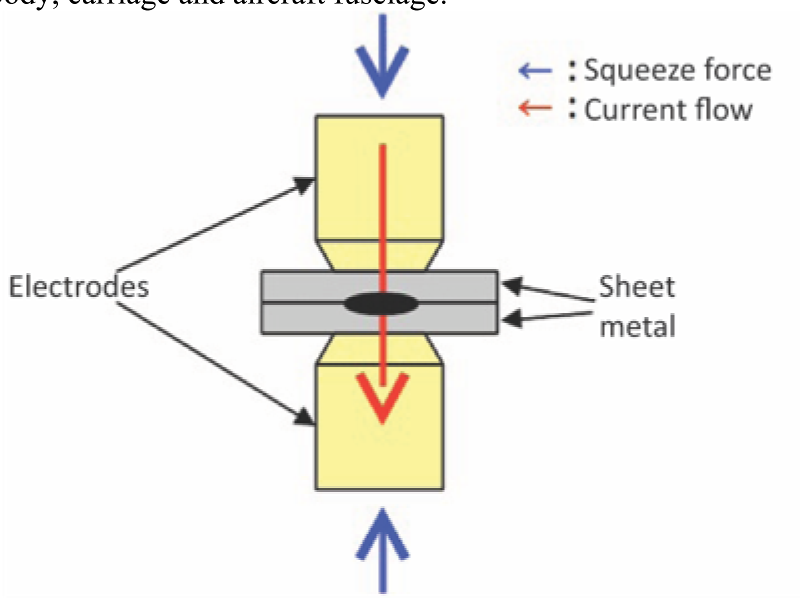


Figure 1.2.1 Spot welding

1.2.2 Seam welding

Seam welding refers to the resistance welding method in which the weldment is assembled into lap joint or butt joint and placed between two roller electrodes, the roller pressurizes the weldment and rotates, continuously or intermittently transmits power to form a continuous weld. The circular electrode is used to replace the cylindrical electrode in spot welding and move relative to the workpiece. It is mainly used for welding of oil barrel, can, automobile oil tank, etc. The welding parts are assembled into lap joint or oblique joint and placed between two roller electrodes. The roller pressurizes the welding parts and rotates, and continuously or intermittently transmits power to form a continuous welding line, which is called seam welding. Seam welding is a kind of welding method in which a pair of roller plate electrodes are used to replace the cylindrical electrodes of spot welding and move relative to the workpiece to produce a sealing weld with overlapping nuggets.

Seam welding is widely used in thin plate welding of oil drums, cans, radiators, aircraft and automobile fuel tanks, as well as sealed containers in jet engines, rockets and missiles.

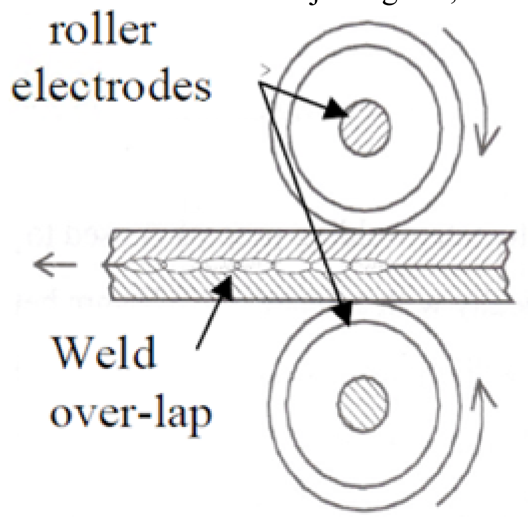


Figure 1.2.2 Seam welding

1.2.3 Butt welding

Butt welding refers to the method that the weldment is respectively placed between two clamping devices to make its end face aligned, and the electric heating is applied at the contact point for welding. For butt welding, the cross-section size and shape of the contact part of the weldment shall be the same or similar, so as to ensure the uniform heating of the contact surface of the weldment.

Butt welding is mainly used for manufacturing closed parts (such as bicycle rim, steel window, etc.); Rolling material lengthening (such as steel town, steel pipe, steel bar, etc.); Welding of dissimilar materials (such as butt welding of dissimilar materials for saving valuable materials and improving the service life of tool working parts). Butt welding is widely used because of its high productivity and easy automation.

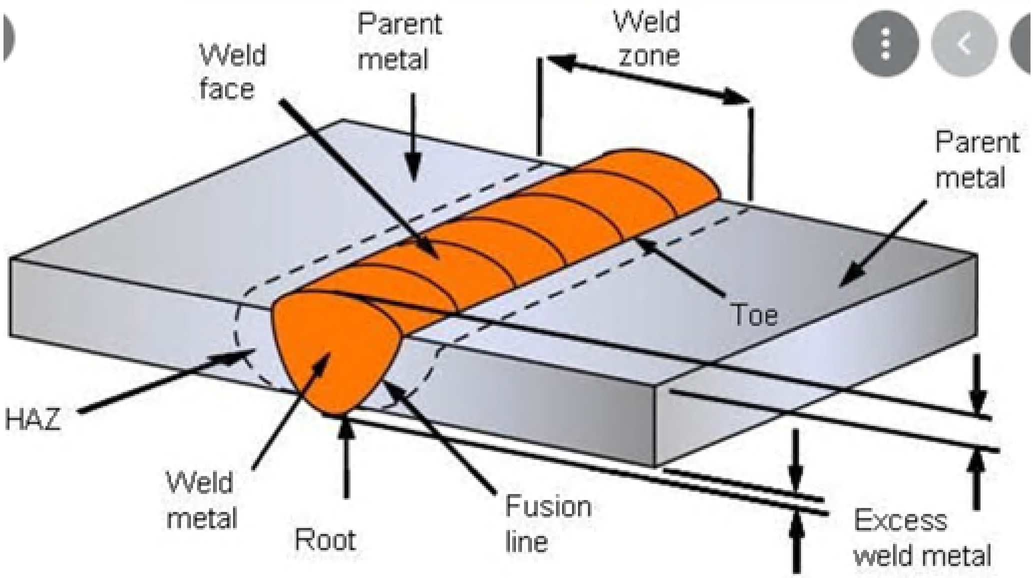


Figure 1.2.3 Butt welding

1.2.4 Projection welding

Projection welding is a kind of high efficiency welding method which can carry out multi-point welding at the same time. It can be used to replace arc welding, brazing and biting. The processing speed of this method is fast, and there is no other consumption except power, which is a great characteristic. Compared with spot welding, the difference of convex welding is that the convex points are machined on the plate in advance, or the surface and chamfer on the weldment which can concentrate the current are used as the contact parts during welding. When welding, the convex point contact improves the pressure and current density per unit area, which is conducive to crushing the oxide film on the surface of the plate, concentrating the heat, reducing the shunt, and reducing the center distance of spot welding.

Multi point convex welding can be carried out at one time, which improves the productivity and reduces the warpage of the joint. On the car body, the convex welding nut (the nut with convex points) is generally welded on the thin plate, so that only the bolts need to be tightened during assembly, which improves the assembly efficiency.

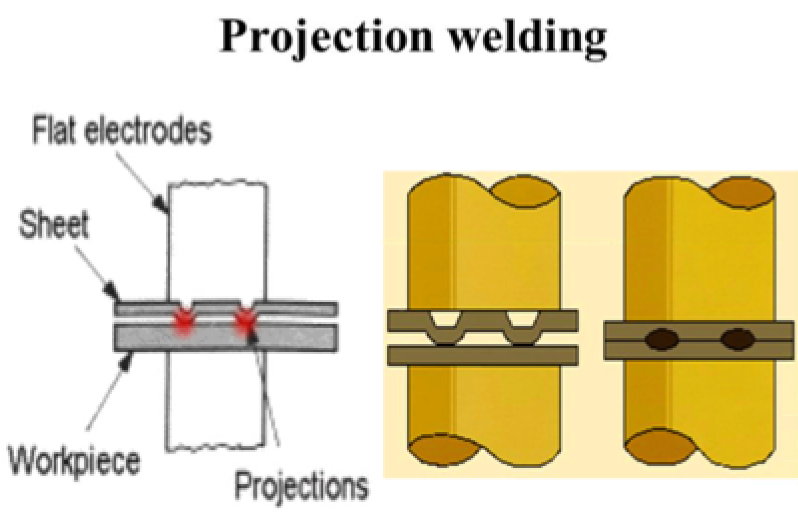


Figure 1.2.4 Projection welding

1.3 Diffusion welding

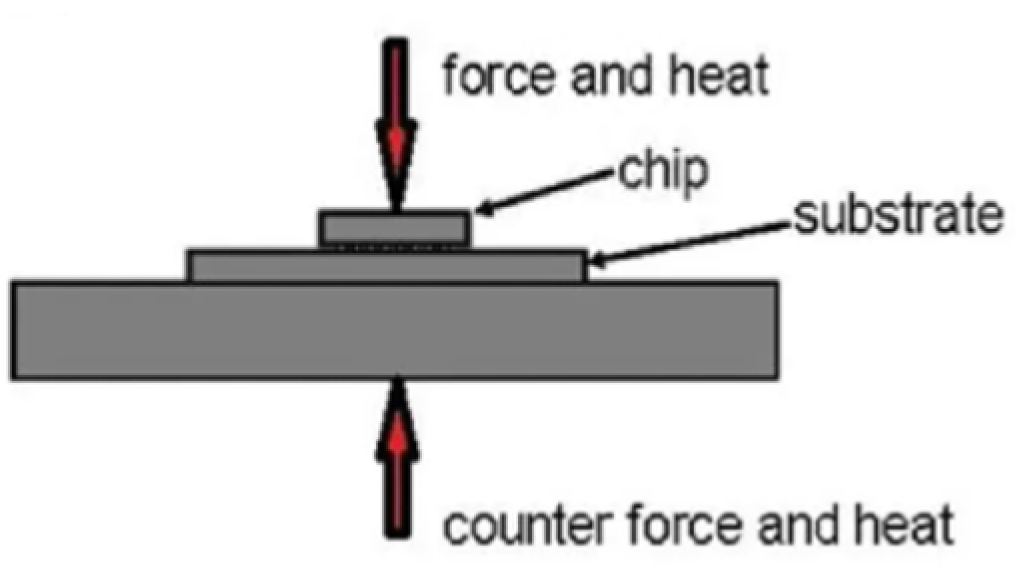


Figure 1.3.1 Diffusion welding

Two surfaces are held together under pressure at an elevated temperature and the parts coalesce by solid-state diffusion. It is characterized by high requirements for the surface quality to be welded, long welding time, and unstable joint quality.

Advantages:

1. Due to the overheating and non-melting of the substrate during diffusion welding, almost all metals or non-metals can be welded without reducing the performance of the welded material. It is especially suitable for fusion welding and other materials that are difficult to weld, such as active metals, Heat-resistant alloys, ceramics and composite materials, etc.

For the same materials with poor plasticity or high melting point, as well as dissimilar materials that are not mutually soluble or will produce brittle intermetallic compounds during fusion welding, diffusion welding is a more suitable welding method.

2. The quality of the diffusion welding joint is good, and its microstructure and performance are close to or the same as the base metal. There are no fusion welding defects in the weld, and there is no overheated structure and heat-affected zone. Welding parameters are easy to accurately control, and the joint quality and performance are stable during mass production.
3. The weldment has high precision and small deformation. Due to the small pressure applied during welding, the workpiece is mostly heated as a whole and cooled with the furnace, so the overall plastic deformation of the weldment is very small, and the workpiece after welding is generally no longer machined.
4. Large cross-section workpieces j I can be welded because the welding pressure is not large, so the tonnage of the equipment required for large cross-section welding is not high and easy to realize.
5. It can weld workpieces with complex structures, inaccessible joints, and large differences in thickness, and can weld many joints in the assembly at the same time.

Disadvantages:

1. The preparation and assembly quality requirement of the surface of the weldment are relatively high, especially for the joint surface.
2. The welding thermal cycle time is long and the productivity is low. Each time welding is as fast as a few minutes and as slow as tens of hours. For some metals, it will cause grain growth.
3. The equipment has a large one-time investment, and the size of the welding workpiece is limited by the equipment, so continuous mass production cannot be carried out.

1.4 Explosion welding

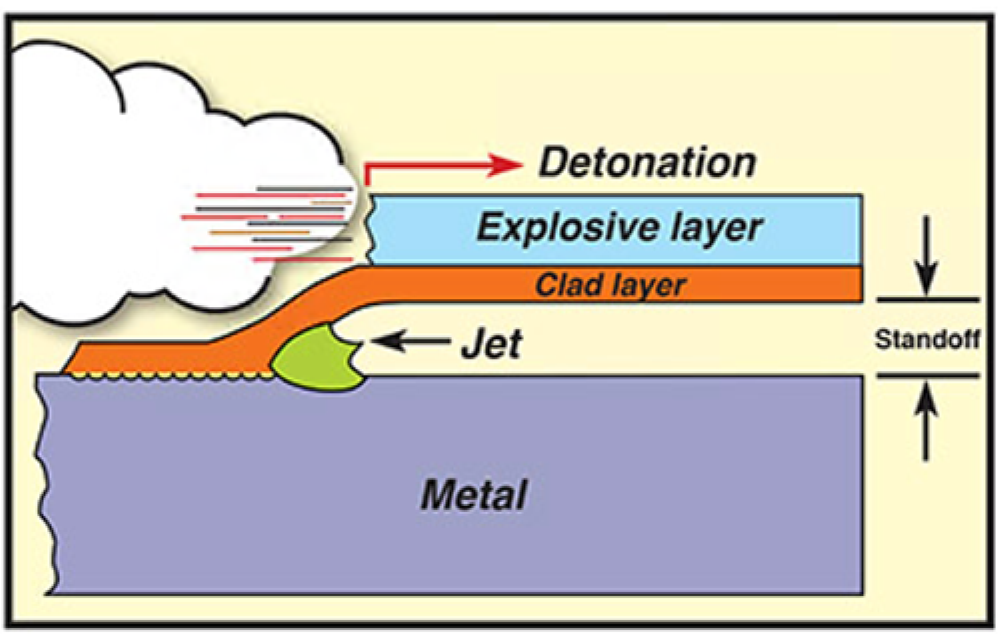


Figure 1.4.1 Explosion welding

Explosive welding is a kind of processing technology in which two dissimilar metal plates are welded together by using the explosive energy. There are two forms of spot welding and wire welding, also known as explosive pressure welding. It is mainly used for welding composite plates of reaction tank and storage tank with little change of internal pressure and temperature.

The application fields of explosive welding and explosive composite materials can be summarized into two main aspects

First, as a new process and technology of metal welding, explosive welding is incomparable to the known welding process and technology. In fact, as long as metal materials have a certain plasticity and impact toughness, they can be explosive welded at room temperature. Even materials with low plasticity and impact toughness can be welded together by thermal explosion.

For the welding of metal materials, the use of new energy will promote the development of welding technology. Explosive welding has become a new welding technology in this field due to the utilization of explosive, a huge energy source.

Second, explosive welding is a new process and technology in the production of metal composite materials, which is its biggest use. Especially after the combination of explosive welding with various pressure processing and machining processes, the production methods and processes of other composite materials will be eclipsed. Explosive welding has obvious advantages in variety, specification, output, quality, market, cost and benefit. Practice has proved that the field of explosive composites is the enrichment and expansion of material science system, explosive composites is a new development direction and one of the frontiers, and it is also an important front army to realize the sustainable development of materials.

1.5 Cold welding

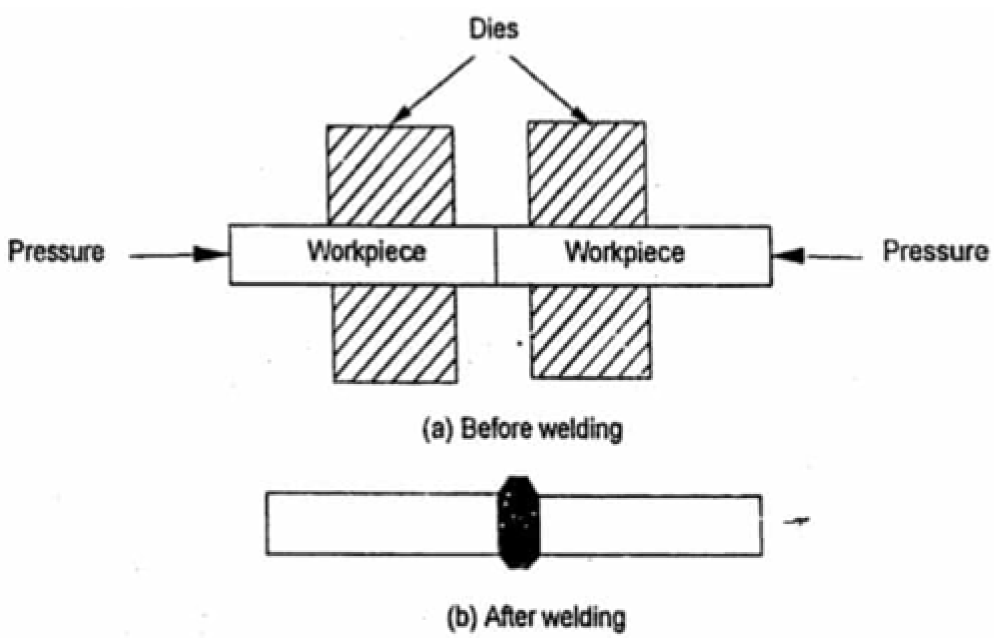


Figure 1.5.1 Cold welding

Cold welding is a kind of method which uses mechanical force, molecular force or electric power to make the welding material diffuse to the surface of the appliance. At the micro scale, this phenomenon is closely related to the diffusion of atoms on the surface of materials.

The cold welding process is carried on the cold welding machine, which is also called ESD. It can be divided into hardfacing cold welding machine, sticking sheet repair cold welding machine and welding copper aluminum wire cold welding machine. The first two are used to repair the wear, scratch, air hole, sand hole and other small defects on the surface of metal and casting; The latter is used for welding copper wire, aluminum wire and other non-ferrous metal wire, such as motor enameled wire.

Due to the difference between cold welding and traditional welding, cold flux has high hardness, adhesion and strength, almost no shrinkage, and can reliably prevent many chemical effects, physical stress and mechanical stress, which is also called "liquid metal". It is one of the main applications of cold welding to connect the same or different materials by using reasonable process and selecting appropriate chemical bonding materials (adhesive, sealant, fixing agent, repair agent, etc.) to realize connection, sealing, fixing and functional coating.

1.6 Friction welding

Friction welding is a kind of welding method which uses the heat generated by the friction of the contact surface of the workpiece as the heat source to make the workpiece produce plastic deformation under the action of pressure.[3]

Under the action of constant or increasing pressure and torque, the friction heat and plastic deformation heat are generated in the friction surface and its adjacent area by using the relative motion between the welding contact end faces, which makes the temperature of the friction surface and its adjacent area rise to a temperature range close to but generally lower than the melting point. The deformation resistance of the material is reduced, the plasticity is improved, and the oxide film on the interface is broken, Under the effect of upsetting pressure, with the plastic deformation and flow of the material, the solid-state welding method is realized through the molecular diffusion and recrystallization of the interface.

Advantages:

1. The welding quality is good and stable. During the welding process, the material to be welded does not melt, and there are no welding defects and joint embrittlement related to melting and solidification. The metal in the welding zone is forged structure. Short welding time and narrow heat affected zone make it easy to obtain good quality welded joint.
2. It can be applied on a wide range of materials. Most of the same and different metals can be welded by friction welding. The dissimilar materials, such as aluminum steel, aluminum copper, titanium copper, intermetallic compound steel, which are difficult to be welded by conventional fusion welding methods, can be welded, and can be used to weld composite materials, functional materials, refractory alloys and other new materials.
3. The welding duration is short with a good productivity. The production efficiency of friction welding is 1 ~ 100 times higher than other welding methods, which is suitable for mass production
4. High precision and low cost for the weldment. The welding deformation caused by welding thermal cycle is small, the dimensional accuracy is high after welding, and there is no need to calibrate and eliminate stress after welding

Disadvantages:

1. The equipment is complex and the cost is huge, so it is only suitable for mass production
2. This welding process is carried out by workpiece rotation which is difficult to weld non-circular cross-section
3. It is difficult to weld the plate-shaped workpiece and thin-walled pipe because these are not easy to clamp
4. This process is limited by the welding machine spindle motor power.

1.7 Ultrasonic welding

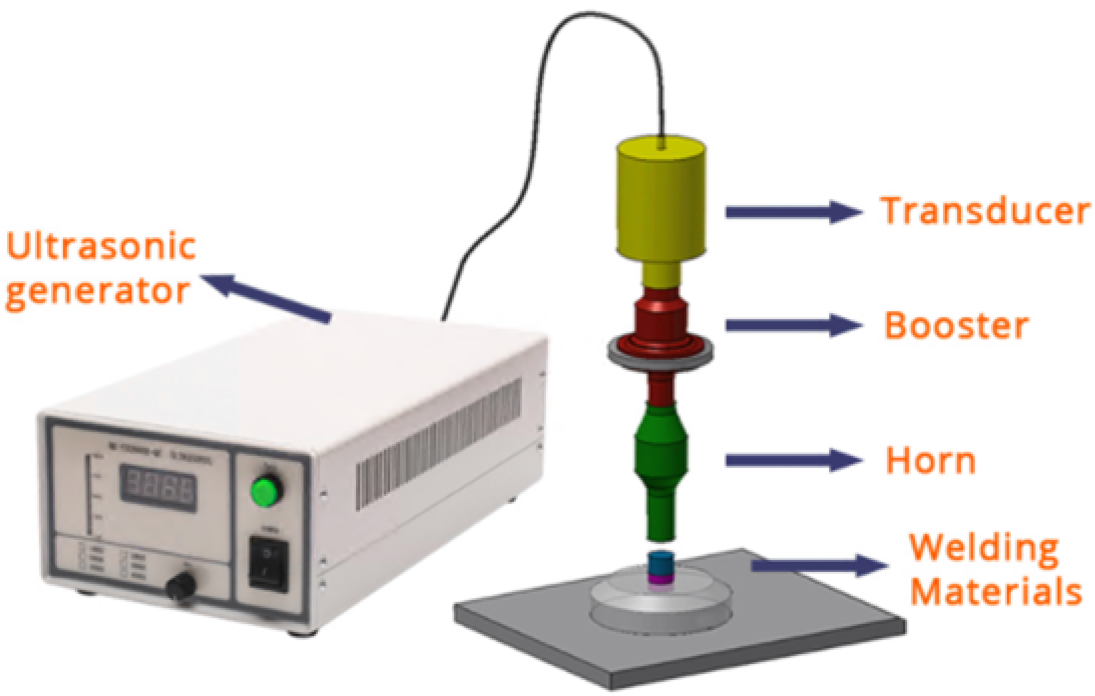


Figure 1.7.1 Ultrasonic welding

The principle of ultrasonic metal welding is using the mechanical vibration energy of ultrasonic frequency (more than 16KHz). It's a special method to connect the same metal or different metal. When the metal is welded by ultrasonic, it neither transmits current to the workpiece nor applies high temperature heat source to the workpiece, but only transforms the vibration energy of wire frame into the friction work between the workpiece under static pressure The metallurgical bonding between joints is a kind of solid-state welding without melting of base metal. Therefore, it effectively overcomes the spatter and oxidation phenomenon during resistance welding. Ultrasonic metal welding machine can carry out single point welding of copper, silver, aluminum, nickel and other non-ferrous wire or sheet materials It can be widely used in the welding of thyristor lead, fuse, electrical lead, lithium battery pole piece and pole lug.

The advantages of ultrasonic metal welding are fast, energy saving, high fusion strength, good conductivity, no spark, close to cold processing; The disadvantages are that the welded metal parts should not be too thick (generally less than or equal to 5mm), the welding spot should not be too large, and the pressure should be applied.

Ultrasonic metal welding is a kind of mechanical treatment process. In the welding process, there is no current flowing through the weldment, and there is no welding arc such as electric welding mode. Because ultrasonic welding does not have the problems of heat conduction and resistivity, it is undoubtedly an ideal metal welding equipment system for non-ferrous metal materials, It can weld effectively.

2 CHAPTER

FRICTION STIR WELDING PROCESS

2.1 History

Friction Stir Welding (FSW) is a non-fusion welding process unlike conventional joining process which are fusion in nature and is derivative of friction welding and produces good quality lap and butt joints. Friction stir welding is a joining process which is developed and patented by TWI Ltd, Cambridge, UK in 1991[4]. TWI filed successfully for patents in Europe, the U.S., Japan, and Australia. TWI then established TWI Group-Sponsored Project 5651,"Development of the New Friction Stir Technique for Welding Aluminum," in 1992 to further study this technique.

The development project was conducted in three phases. Phase I proved FSW to be a realistic and practical welding technique, while at the same time addressing the welding of 6000 series aluminum alloys. Phase II successfully examined the welding of aerospace and ship aluminum alloys, 2000 and 5000 series, respectively. Process parameter tolerances, metallurgical characteristics, and mechanical properties for these materials were established. Phase III developed pertinent data for further industrialization of FSW.

Since its invention, the process has received world-wide attention, and today FSW is used in research and production in many sectors, including aerospace, automotive, railway, shipbuilding, electronic housings, coolers, heat exchangers, and nuclear waste containers.[5]

FSW has been proven to be an effective process for welding aluminum, brass, copper, and other low-melting-temperature materials. The latest phase in FSW research has been aimed at expanding the usefulness of this procedure in high-melting-temperature materials, such as carbon and stainless steels and nickel-based alloys, by developing tools that can withstand the high temperatures and pressures needed to effectively join these materials.

2.2 The definition and principle of FSW

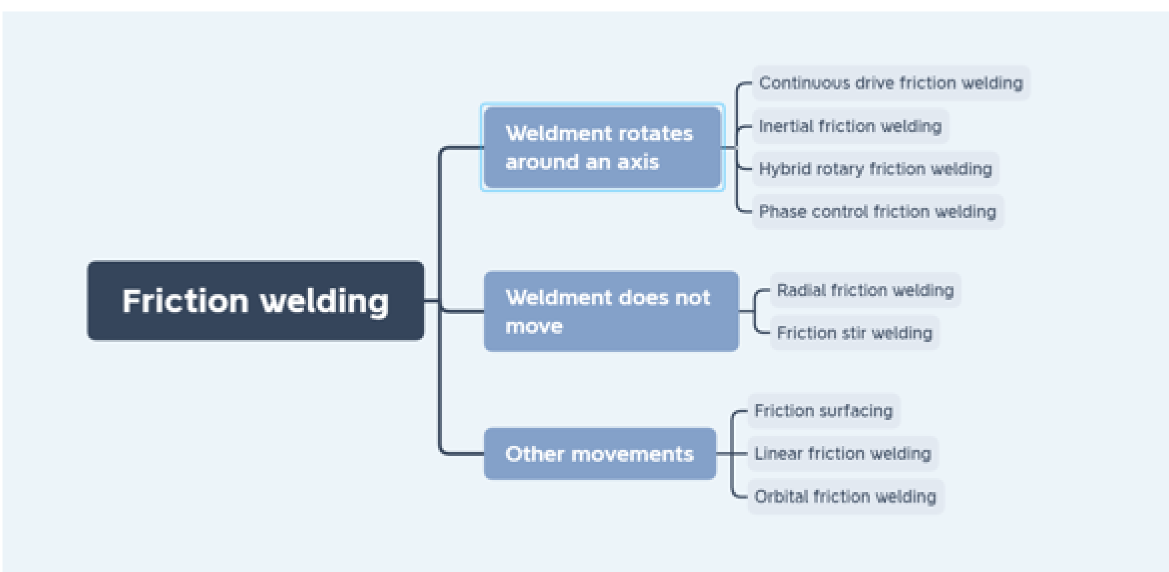


Figure 2.2.1 Classification of friction welding

Friction stir welding method is the same as conventional friction welding, friction heat is also used as welding heat source in friction stir welding. The difference is that the welding process of FSW is completed by a cylindrical welding head (extending into the joint of the workpiece) which rubs against the material of the workpiece through the high-speed rotation of the welding head, so as to increase the temperature of the material at the joint and soften it. At the same time, the material is stirred and rubbed to complete the welding.

FSW joins materials through the use of a non-consumable rotating tool. A typical FSW tool is comprised of two main components: the shoulder and the pin. The rotating tool is plunged at the start of the joint until it is at the appropriate depth where the shoulder contacts the material. The tool then traverses along the weld joint while continuing to rotate. After traveling the desired length, the tool is retracted.

During welding, the material is heated by a combination of frictional and adiabatic heating, plasticizing the base material. This plasticized material is rotated around the tool and consolidated due to compressive forces from the tool.[6] Due to the rotation rate and compressive forces applied by the tool, the base material experiences a high strain rate which promotes dynamic recrystallization. This phenomenon leads to a very fine grain microstructure.

2.3 Friction Stir Welding Process

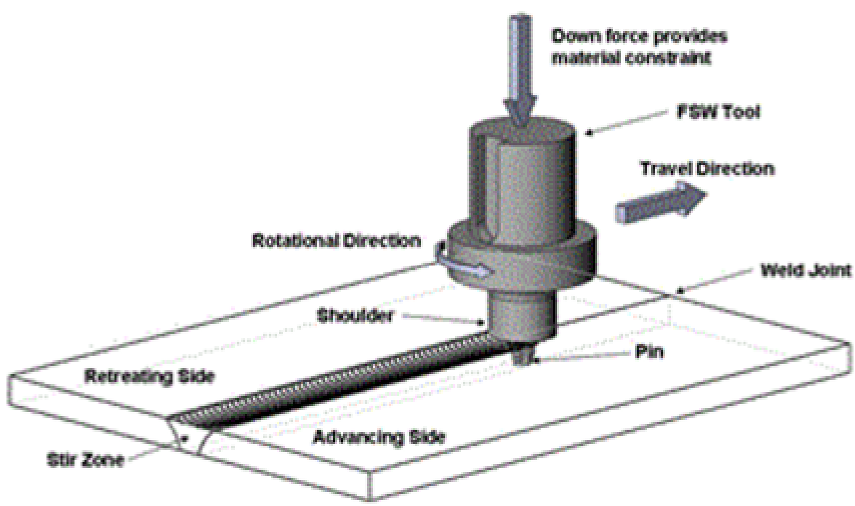


Figure 2.3.1 Friction Stir Welding process

Friction stir welding (FSW) is a kind of solid phase bonding method using friction heat as welding heat source, but it is different from conventional friction welding. During friction stir welding, the weldment is firmly fixed on the working platform. Then, the stir welding head rotates at high speed and the stir welding pin is inserted into the joint of the weldment until the shoulder of the stir welding head is in close contact with the surface of the weldment, The heat generated by the friction between the welding pin and its surrounding base metal and the friction between the shoulder of the welding head and the surface of the weldment will increase the temperature of the material at the joint and soften it.[7] At the same time, the rotating edge of the welding head moves relative to the weldment along the joint, resulting in strong plastic deformation of the material in front of the welding head. As the head moves forward, the material with high plastic deformation at the leading edge is squeezed behind the head. A compact solid-phase joint was formed by friction heat generation and forging between the shoulder of the stir head and the surface of the weldment.

2.3.1 Joint types

Friction stir welding can realize the reliable connection between bar and bar, plate and plate. The joints can be designed as butt joint, lap joint, fillet joint and T-joint. It can be used for the welding of circular, circular, nonlinear and three-dimensional welds. Since gravity has no effect on this solid-state welding method, friction stir welding can be used for all position welding, such as horizontal welding, vertical welding, overhead welding, annular rail automatic welding, etc.

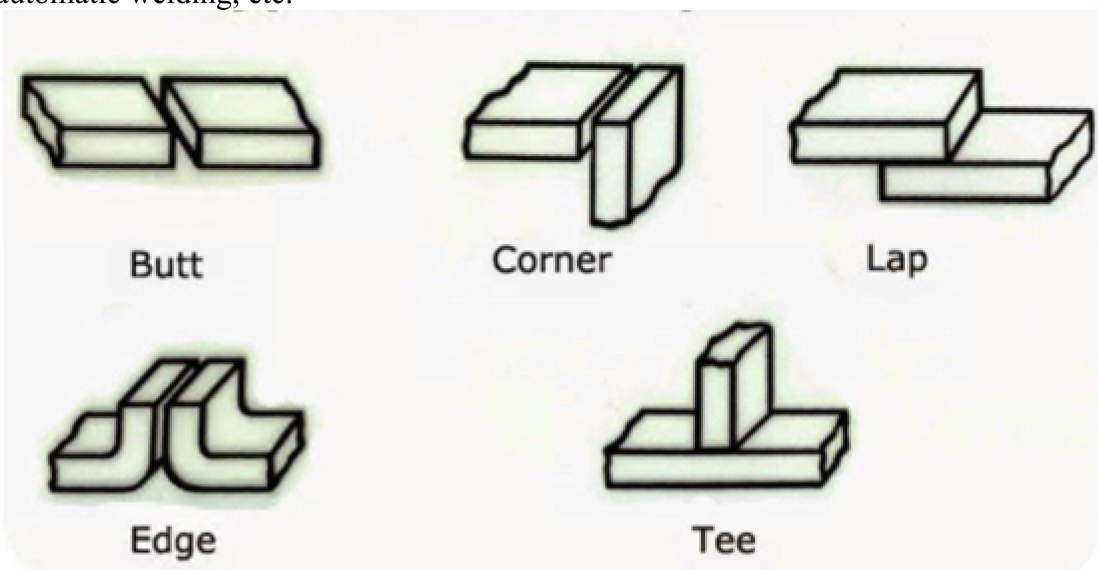


Figure 2.3.1 Types of joints

2.3.2 Heat input of friction stir welding

In the process of friction stir welding, the pin rotates at high speed and inserts into the weldment. Then, under the action of welding pressure, the shoulder contacts the surface of the weldment. As a result, a large amount of friction heat is generated between the shoulder and the upper surface of the weldment and between the pin and the joint surface. At the same time, plastic deformation and fluid flow occur near the pin, resulting in deformation and heat generation, Friction heat is the main part of welding heat. As the welding head moves along the weld direction, the heat exerts thermal cycling on the weld and the base metal near the weld, resulting in the dissolution of precipitates in the material and the softening of the weld and heat affected zone. [8]Friction stir welding is essentially a welding method with friction heat as the welding heat source, so the heat input is a direct and key factor affecting the welding quality.

There is an optimum range between the temperature in the welding seam and the mechanical properties of the joint. If it is beyond the optimum range, the heat in and out of the welding seam is too large, and the mechanical properties of the joint will be reduced. Reason: in the welding process of aluminum alloy, the heat effect zone (HAZ) where the microstructure and properties change on both sides of the welding seam due to the thermal cycle is the main dangerous zone for softening. The width of the softening zone is directly proportional to the heat input, When the temperature in the weld enters the softening temperature of the aluminum alloy, the precipitation and aggregation of strengthening phase will occur in the heat affected zone, the solution strengthening effect of the material will be weakened, and the strength of the weldment will be reduced. With the increase of temperature, the strengthening phase will even occur over aging precipitation, the solution strengthening effect of the material will be worse, and the strength will decrease more.

2.3.3 **Welding parameters**

Friction stir welding is a complex process. On the premise of the determination of the FSW head, the most important process parameters of FSW are the rotation speed R, welding speed V of the FSW head and the down pressure of the shaft shoulder. The forming characteristics and properties of friction stir welded joints are closely related to the energy absorbed per unit length of weld in the process of friction stir welding. The energy absorbed per unit length of weld is related to the ratio R/V of rotation speed to welding speed. The larger R/V value is, the more times the stir head rotates on the weld with unit length, the more heat input into the material, The higher the temperature of the welding zone is.[9] On the contrary, the lower the R/V value is, the lower the temperature is.

2.4 The characteristics of FSW

Compared with traditional friction welding and other welding methods, friction stir welding has the following advantages[10]

1. The quality of welded joint is high, and it is not easy to produce defects. The weld is extruded in the plastic state and belongs to solid-state welding. Therefore, the joint will not produce some welding defects and embrittlement phenomena related to solidification metallurgy, such as cracks, pores and burning loss of alloy elements. It is suitable for welding colored metals such as aluminum, copper, lead, titanium, zinc, magnesium and their alloys, as well as steel materials and composite materials, It can also be used to connect different materials.
2. Not limited by the shaft parts, it can be used for flat plate butt joint and lap joint, straight weld, fillet weld and girth weld, large frame structure and large cylinder manufacturing, large flat plate butt joint, etc., expanding the scope of application.
3. Easy to realize mechanization and automation, stable quality and high repeatability. Friction stir welding process parameters less, simple welding equipment, easy to achieve automation, so that the welding operation is very simple, the reliability of welding machine operation and welding quality is greatly improved.
4. Low welding cost and high efficiency. There is no need to fill material and protective gas, no need to pre treat the surface of weldment before welding, and no need to apply protective measures during welding. The edge of thick weldment need not be beveled. It is not necessary to remove the oxide film for welding aluminum workpiece, just remove the oil stain. It is allowed to leave a certain gap during butt joint, and the assembly accuracy is not required.
5. Small welding deformation and high dimensional accuracy of weldment. Because friction stir welding is a solid-state welding, its heating process has the characteristics of high

energy density and fast heat input, so the welding deformation is small and the residual stress is small. Under the conditions of enough rigidity of welding equipment, accurate assembly and positioning of weldment and strict control of welding parameters, the dimensional accuracy of weldment is high.

6. Green welding. There is no arc radiation, smoke and spatter in the welding process, and the noise is low. Therefore, friction stir welding is a "green welding method" with high quality and low cost.

At the same time, friction stir welding also has some shortages

1. The design, process parameters and mechanical properties of welding tools are only applicable to a small range of alloys with a certain thickness.
2. The wear of the stir welding head is relatively high.
3. At present, the welding speed is not high.
4. Special fixture is needed, and the flexibility of the equipment is poor.

2.5 Applications

1. In the aerospace field, friction stir welding has been successfully applied to the welding of longitudinal butt weld and circumferential lap joint of rocket and space shuttle booster fuel cylinder. The 2014-T6 aluminum alloy with a diameter of 2.4m and a thickness of 22.2mm was welded with a friction stir welding machine called superstir produced by ESAB company δ Compared with MIG welding, the defect rate of longitudinal seam of rocket fuel cylinder is very low. There is a defect in the length of MIG welding seam 832cm, while there is a defect in the length of friction stir welding seam 7620cm, which is 1/10 of that of MIG welding. Recently in δ IV rocket. There is no defect in the 1200m long weld of friction stir welding.[11]

2. In the electronic industry, friction stir welding has been used in the welding of large aluminum alloy radiator. Large wide aluminum alloy radiator is an important part of high-speed trains, urban rail and other rail vehicles.

The melting point and thermal conductivity of copper are higher than that of aluminum, so it is very difficult for copper and copper alloy to adopt general fusion welding method. In Europe, a large copper vessel was made by friction stir welding to store high-energy radioactive materials. The cover and the cylinder were welded together. The penetration was 58mm, the shoulder diameter of the stirring head was 60mm, and the temperature near the joint was as high as 750 ℃. Compared with non vacuum electron beam welding, the welding speed of non vacuum electron beam welding is 254 mm/min, while that of friction stir welding is 100.4 mm/min. the welding speed is slower, but the heating input is not increased, so it can meet the requirements.[12]

3. As a mature technology, friction stir welding has been applied to the flat butt joint of Mercedes Benz SL aluminum body and the butt joint of continuous curved surface parts with small curvature radius.

Energy saving and environmental protection is the general trend of automobile manufacturing industry. The most intuitive way to solve the high emission of automobile is to lighten the body. More than 98% of Mercedes Benz SL body parts are made of aluminum. In addition to mechanical connection, TIG fusion welding is also used. Friction stir welding technology is used on the main floor of the body.[13] Among them, weld a and weld B are butt joints of flat plate, the thickness of plate is consistent, the distribution of fish scale is very uniform, and the back weld mark is very stable, that is, the parameters of connection process are consistent.

With the improvement of people's understanding of friction stir welding technology, it is expected that in the near future, the connection of aluminum alloy materials will be mainly completed by friction stir welding, especially in the launch vehicle, high-speed aluminum alloy train, aluminum alloy high-speed boat, all aluminum alloy automobile and other projects, friction stir welding technology will play a leading role

3 CHAPTER

SIMULATION METHODS OF FSW PROCESS

3.1 Introduction

3.1.1 History of the simulation of welding temperature field

The essence of welding process is that the material is heated and melted under the action of external heat source, and then solidified under certain cooling conditions. In this process, the change of temperature is the most important factor affecting the welding forming and welding performance. Therefore, it is very important to understand the law of temperature change of material under the action of heat source for studying the welding method and process of material.

As early as the early 20th century, scientists began to study the welding temperature field, and the initial research direction was only limited to the heat conduction law in solid materials.[14] In 1935, D. Rosenthal studied the heat transfer mechanism of moving heat source in solid by using the differential equation of heat conduction.

In the 1940s, D. Rosenthal and H. Rykalin obtained the rekalin formula to describe the welding temperature field by simplifying the welding model and using the analytical method. Limited by the scientific research conditions at that time, the material phase transition and a series of other physical and chemical changes involved in the temperature change process were not included in the scope of the model study, The analytical results are often far from the actual values. After that, researchers have done a lot of research and modification on Rosenthal rykalin model[15], which greatly improves the calculation accuracy of welding heat source model. After a lot of experiments, ADAMES, Muyuanbo and Daogeng Daofu have accumulated the measurement data under different parameters and established the welding heat conduction equation under different conditions. Although the workload of this method is large, the results are more accurate than the mathematical analysis method. With the rapid development of computer science, the computing power of computer has far exceeded the limit of human computing power at the beginning of the century. Wilson and Nikcell began to use the finite element method to calculate the parameters of solid heat transfer process.

In 1975, Poley first used computer program to simulate the change of welding temperature field. In 1976, Krutz's paper discussed the possibility of predicting weldability by thermal cycle of temperature field. In the two-dimensional temperature field model, it is assumed that the forward speed of heat source is faster than the heat conduction speed of base metal, and the influence of heat conduction in the forward direction of heat source is ignored in the process of welding simulation. Since then, J. Goldak proposed a new heat source model based on Gauss heat source for arc welding and submerged arc welding. This model considers that the welding energy obeys Gauss distribution, and further studies the flow of molten liquid in the welding pool, so as to make the energy distribution of weld more reasonable in the welding simulation process, and make the welding current, voltage The welding parameters such as welding speed are better connected with the simulation parameters, and the simulation results are closer to the actual welding process. With the continuous improvement of computer performance, the heat source model has gradually developed from two-dimensional to three-dimensional. U. Dilthey et al. established a three-dimensional heat source model for GMAW of thin plate, and studied the influence of thermodynamics, hydrodynamics and other factors on the welding process. After entering the 1990s, with the development of finite element software, the welding simulation technology gradually matured. Kamel abderrazak and others used the general finite element software to simulate the welding of magnesium alloy. In 2000, Ronda et al. From the University of Cape Town deduced the phase transformation law and phase transformation plasticity, established the TMM model and formed a unified system. The model not only studies the effect of temperature field on phase transformation, but also studies the effect of stress and strain on phase transformation, which makes the finite element model for the simulation of welding process more mature[16-18]

3.1.2 **Different types of computation methods**

Due to the need of engineering application, researchers have done a lot of research work. At present, the commonly used solving methods can be divided into three categories: empirical method, analytical method and numerical simulation method.[19,20]

1. Empirical method

In the early stage of empirical method, the prediction of welding temperature field and welding stress field was mainly based on experience and experiment. Through the accumulation of a large number of data, the rule of temperature change and welding deformation were predicted. The accumulation of empirical formula is limited by the use conditions, which can simulate the welding of simple structure to a certain extent. However, in the actual production process, the welding structure is not only composed of simple welding seams. The actual welding seam is often with many passes, many layers and complex joint distribution, so this method is difficult to meet the actual needs.

2. Analytical method.

The analytic method is to form mathematical equation model to describe the actual welding process by the derivation of existing laws and laws. By solving the equation, the analytical solution of the relevant welding model can be obtained. Compared with the empirical method. The physical concept of this method is clear and the logic is rigorous; However, due to the fact that most of the conditions are hypothetical and different from the actual situation, the scope of application is limited, so it is difficult to combine with the actual conditions. For some complex high-order, nonlinear equations, it is often difficult to get the required solution.

3. Numerical simulation.

The analytical solution of welding model is obtained by calculating the software. This method is called numerical simulation method. At present, the commonly used welding numerical simulation methods include: Monte Carlo method, numerical integration method, finite element difference method and finite element method, in which the finite element method has the advantages of high accuracy and high efficiency, which has been developed into the main research means of welding numerical simulation. For the finite element analysis of welding stress and deformation, the more commonly used methods are inherent strain method and thermoelastic-plastic method.

3.1.3 **Heat source simulation**

The temperature of welding process changes with time, and the temperature of different areas of base metal is different, which forms the temperature field. Finite element analysis is an important method of welding temperature field simulation. The microstructure and properties, metallurgical process and solid phase transformation in welding process are closely related to heat input and heat conduction. The welding process is a short and fast process, which is usually described as a transient process. The temperature of each area of the welded joint increases rapidly and then decreases rapidly. With the movement of the welding heat source, the temperature field near the heat source changes rapidly with the welding time and relative space position, and some physical and chemical properties of the material itself will also change with the change. This change not only occurs in the corresponding properties, but also produces complex metallurgical phenomena such as metal melting and latent heat of phase transformation at the joint. All these bring difficulties to the numerical simulation of welding process. In order to ensure the correctness of the welding simulation results, the setting of boundary conditions such as the properties of welding materials, the similarity between the welding heat source model and the actual heat source must be consistent with the actual situation.

The establishment of heat source model is an important link in the simulation of welding temperature field. For different welding methods and different heat input, only using the appropriate heat source model can correctly simulate the results consistent with the actual. Although there are many kinds of heat source models, there are three commonly used models: 2D Gaussian model, 3D Gaussian model and double ellipsoid heat source model. In some special cases, due to the actual needs, the combination of the two heat source models will also be used.

The choice of heat source model mainly depends on the welding method and the heat input in the welding process. For surface heat treatment or butt weld with thin plate thickness or material surface heat treatment, 2-gauss model is usually selected. For larger plate thickness direction, 3D Gauss heat source is usually used. However, no matter which kind of Gaussian heat source can not well represent the characteristics of heat source movement, the movement of heat source will lead to uneven heat distribution. If the Gaussian model is still used under the condition of uneven heat distribution, the final result will have a large error with the actual situation, so when the heat source is moving, it is better to use the double ellipsoid model, This is the reason why speed affects the choice of heat source. If the moving speed is very fast, the rising speed of temperature will be much faster than the falling speed, and the heat distribution before and after will be uniform, so the double ellipsoid model must be selected. The heat source model of double ellipsoid is a kind of model with asymmetric front and back distribution. The proportion of heat input is determined according to the size ratio of the front and back ellipses, which can well reflect the phenomenon of uneven heat distribution.

3.2 Finite element method software introduction

The significance of numerical simulation of welding process is to optimize the structure design and process design by calculating the actual situation of repeated welding process, reduce the workload of welding workers, and improve the welding efficiency and welding quality. However, the actual process of welding process is very complex, so the physical and chemical changes of welding process should be fully considered in the numerical simulation.

For example: thermal cycle process in welding process, physical and chemical change process in molten pool transition, behavior of molten pool flow, mechanical behavior of materials, crystallization and phase transformation of materials, growth of grains, segregation of elements, generation and propagation of cracks and many other conditions. In the decades of welding simulation development, scientists have established many welding models. One of the difficulties brought by the gradual improvement of the models is the improvement of the amount of calculation. With the development of computer technology, a large number of commercial finite element simulation software is becoming more and more mature. The commonly used finite element analysis software includes: NASTRAN, ADINA, fluent, ABAQUS, ANSYS, etc. In addition, there are some special software for welding simulation, such as SYSWELD in France, hearts and QUICK WELDER in Japan. In addition, MATLAB and other software can also do a lot of welding numerical simulation work.[21]

The original research of SYSWELD software came from the welding process simulation of nuclear industry. JB. LeBron, a Frenchman, studied the phase transformation of steel during plastic deformation theoretically and numerically. In 1980, French ESI company and famaton company jointly developed SYSWELD software, which further expanded the application scope of SYSWELD software and rapidly promoted its application in aerospace, automobile industry, shipbuilding and other heavy industry fields. SYSWELD software fully realizes the coupling calculation of mechanical process, heat conduction process and metal metallurgy process, and fully considers the metallographic transformation, latent heat of metallographic transformation and metallographic structure change.

3.2.1 **Process topology**

Here we choose 3 different simulation softwares to compare their process of the simulation:

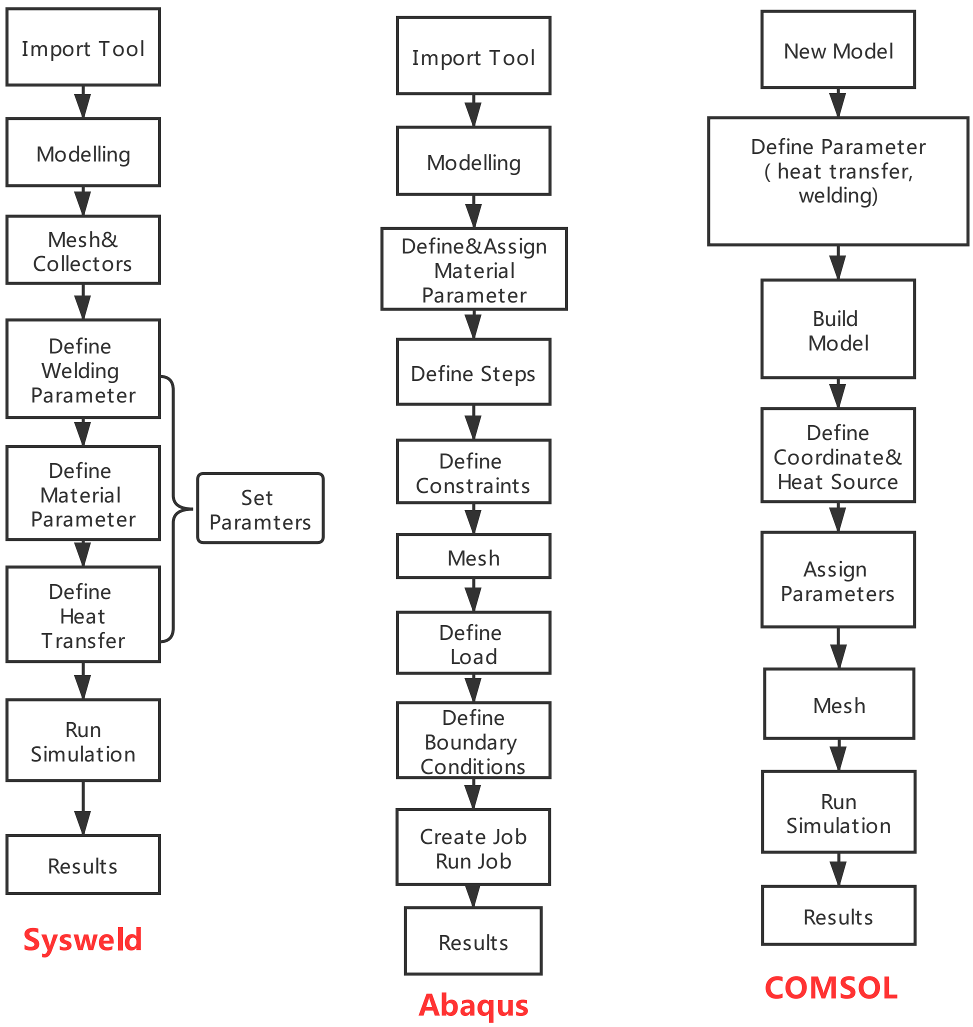


Figure 3.2.1 Operation sequence of simulation

3.3 Detailed comparision

Regardless of the sequence of the simulation process. All these 3 processes are composed by modelling, mesh, parameters and run simulation&check results. I'll discuss about the differences in these domains.

The process in Sysweld and Abaqus is to simulate the real situation of the friction stir welding process. While in COMSOL, the process is simplified. The model is built as a fixed heat source and moving coordinate. And the welding path is considered as infinite, which ignores the computation around the edges of the sheet.

3.3.1 Modelling process

In Sysweld and Abaqus, the user should import or create the model of tool. While in COMSOL, there is no graphical representation of the tool. The computation only focuses on the pin area and shoulder area affected by the tool.

In COSMOL and Abaqus, the model is made of sheets and tool. While in Sysweld, we need also consider the support elements for the welding process.

In Sysweld and COSMOL, the model include the pin area and melting area(shoulder in COSMOL), which is iconical representation of friction stir welding process. While in Abaqus, you just need to define the face contact between tool shoulder and sheet top face, the pin area overlaps with sheet seems to be fine.

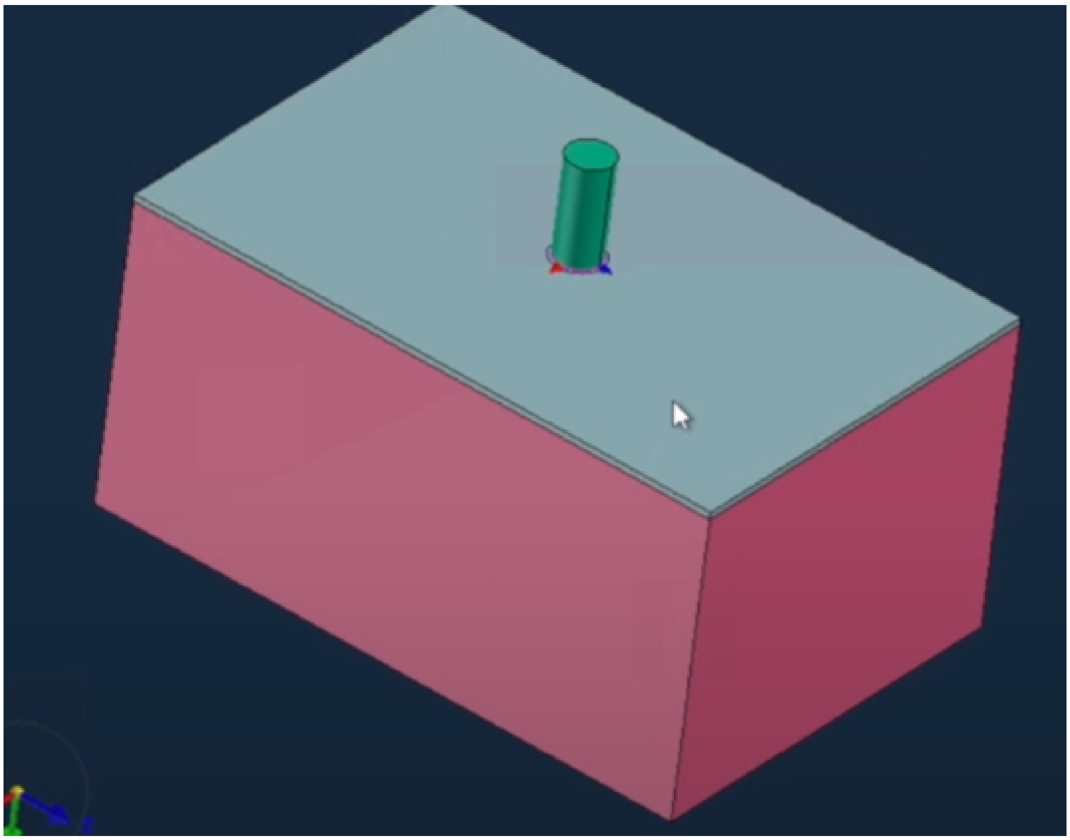


Figure 3.3.1 Model in Sysweld

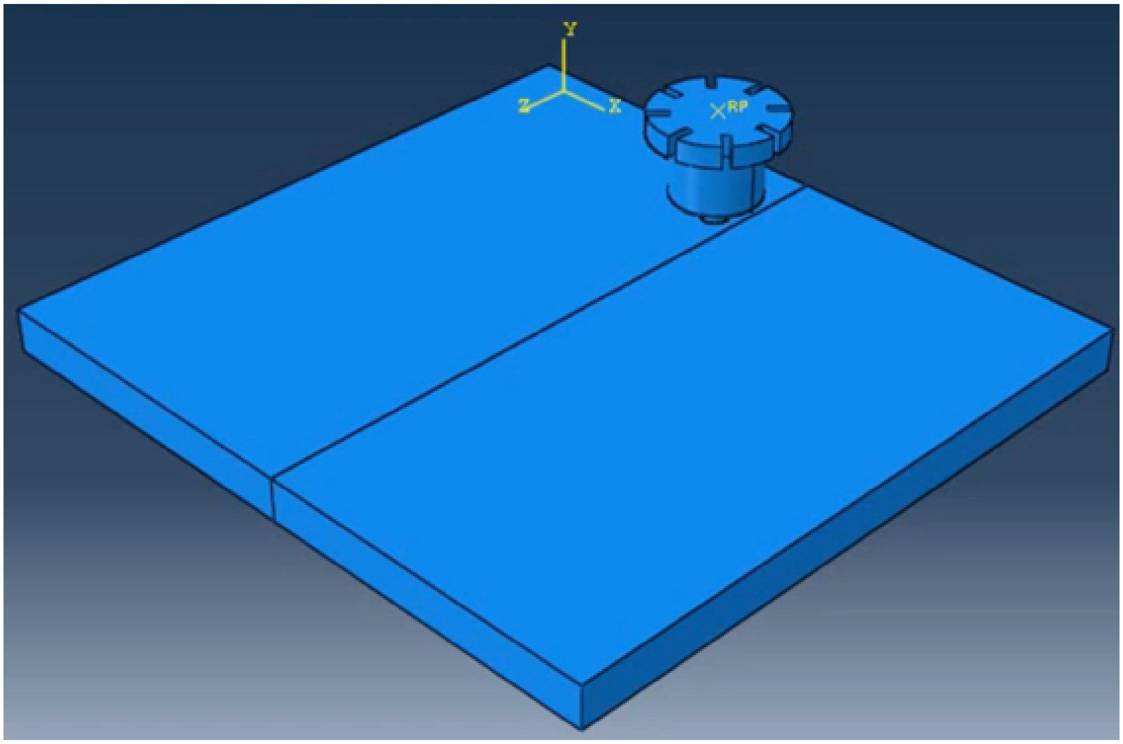


Figure 3.3.2 Model in Abaqus

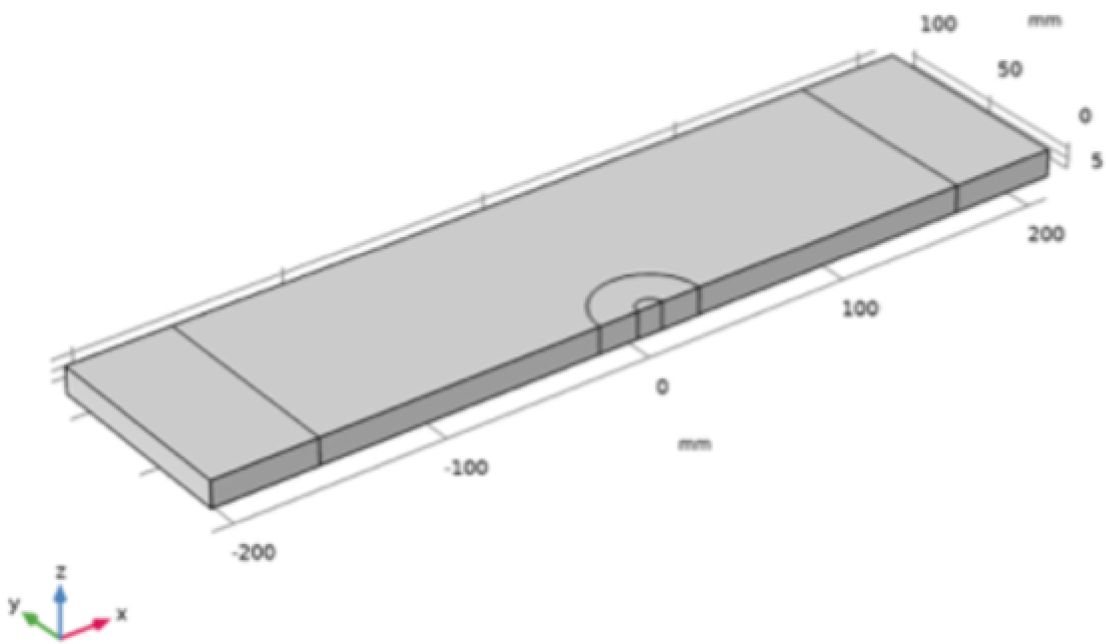


Figure 3.3.3 Model in COSMOL

3.3.2 **Mesh process**

Mesh is the essential procedure in FEM software to run the simulation. The mesh for these 3 methods are more or less the same.

In Sysweld, the mesh is more specific. The user should define the pin and fluid part and name it in collectors. All the boundary conditions, heat transfer and welding parameters are defined by collectors. In this case, the number of collectors are large and all these should be done in mesh procedure which requires the user to have a very good understanding and be very familiar with the process.

In Abaqus, the mesh process is operating contemporarily with the parameter setting process. In this phase, the user is just demanded to mesh the tool and sheet parts. Within parameter setting process, the user will assign the pre-defined material properties to the obtained mesh elements.

In COSMOL, the mesh process is the last before running the simulation. The process is similiar with Abaqus, the user only needs to finish the mesh and assign the material to mesh elements.

3.3.3 **Parameters**

In parameters sets phase, Sysweld will be the most complex and less user-friendly software.

For Sysweld, there is no graphical representation for the user to have a quick understand of the title of input box. User should deal with two document file to filling in the parameters related to welding process, heat transfer and material properties. The only benefit from this method is that as long as the user is fluently grasp the simulation process, he can modify the parameters in a very short time and very effieciently to start a new simulation.

For Abaqus, the parameter setting process is operating contemporarily with the mesh process.

As long as you open the software, it will ask you to fill in the basic numbers to make sure what system of units you are using. And the software has a library to record the material properties you are using. (By user input.) After you create your mesh, you can directly assign the elements with the material in your library. Different from Sysweld, there are no collectors in this software. But more in detail, the software need you to define the boundary conditions, set steps which define the start of the simulation and end of the simulation, and apply load. The user will have a clear view of their inputs and this is easier for beginner to understand.

While in COSMOL, the process is very similar with Abaqus. The user should define the basic parameters, material properties, define steps and boundary conditions.

3.3.4 **Simulation and results**

The simulation is carried on after the parameter are set.

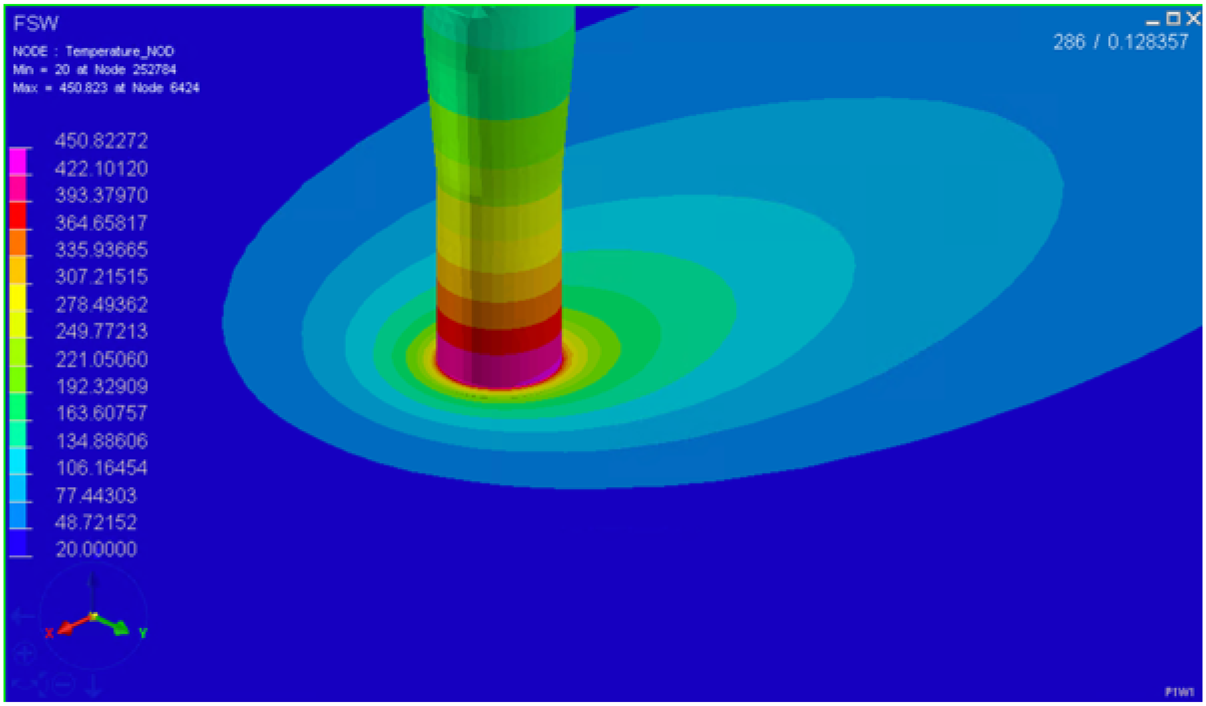


Figure 3.3.4 Results in Sysweld

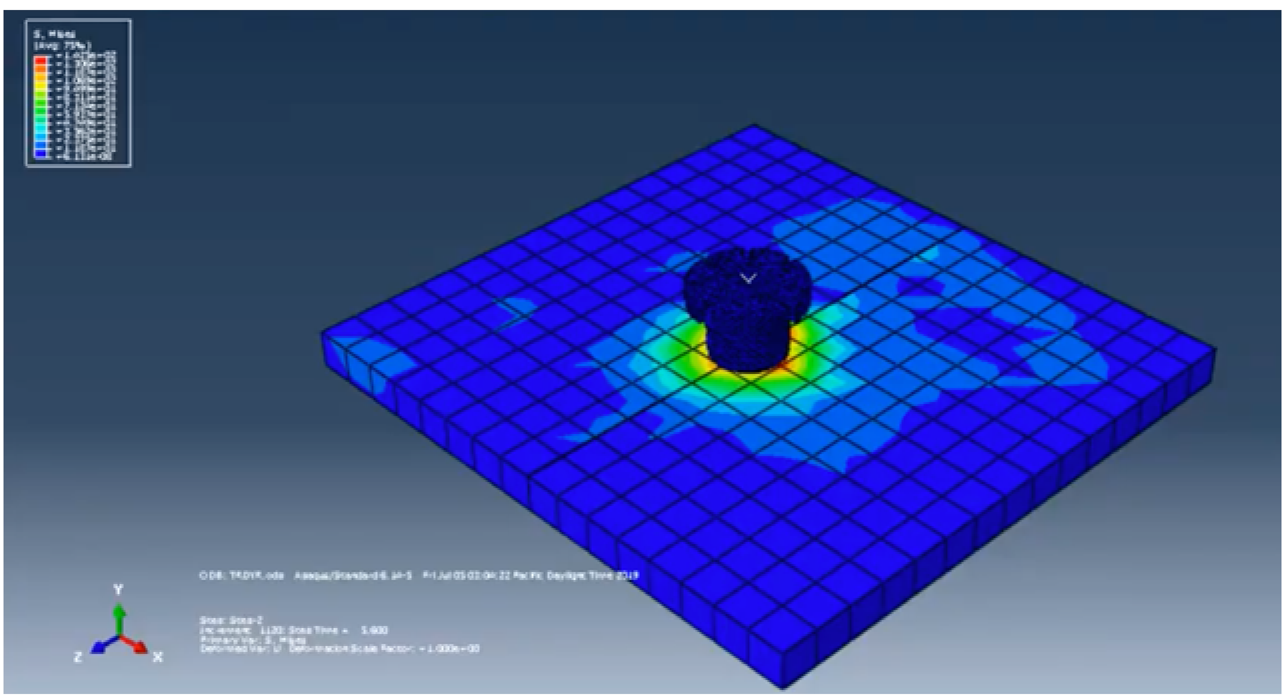


Figure 3.3.5 Results in Abaqus

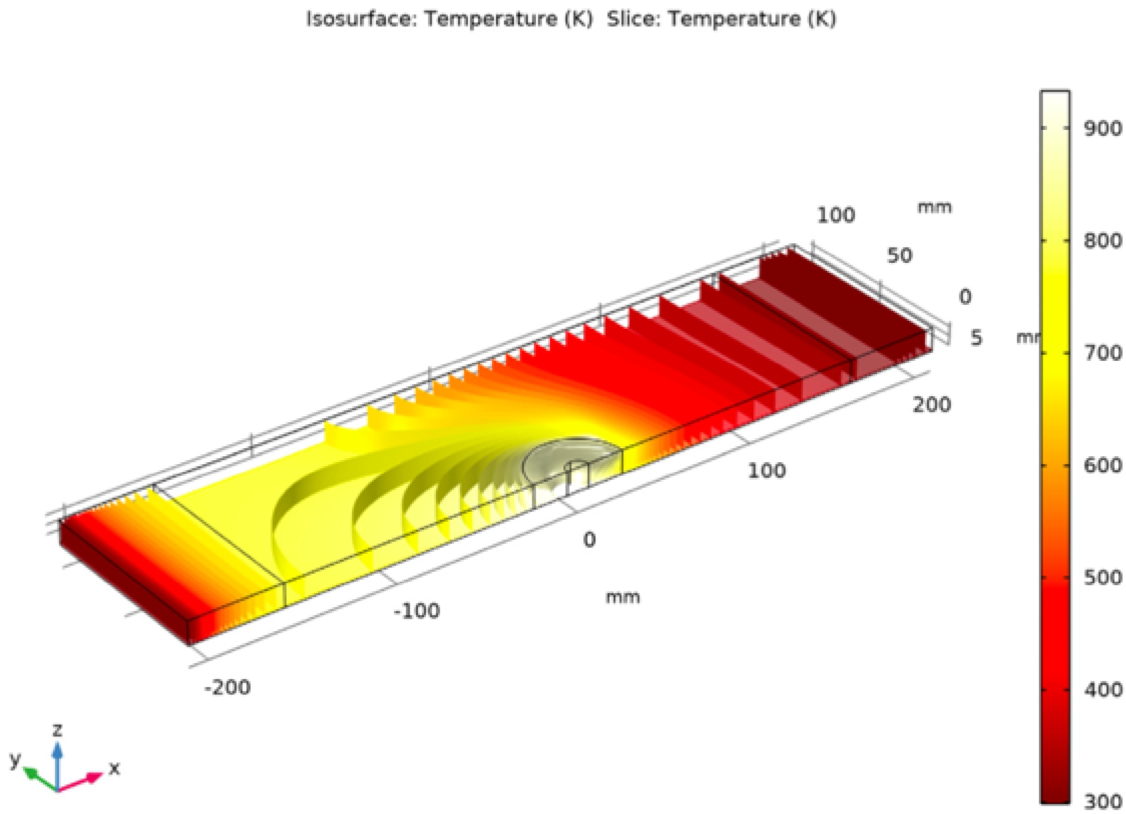


Figure 3.3.6 Results in COSMOL

3.3.5 **Summary**

The process in Sysweld and Abaqus is to simulate the real situation of the friction stir welding process. While in COMSOL, the process is simplified. The model is built as a fixed heat source and moving coordinate. And the welding path is considered as infinite, which ignores the computation around the edges of the sheet.

In terms of the workload, Sysweld needs the most of work due to a huge number of collectors in the mesh phase. But in Abaqus, with no defination of pin and fluid area in the mesh, probably the computation result will be less accurate.

Considering the re-usability, in Sysweld, the user can quickly modify the parameters and get a new simulation result if the material is changed for example. While in Abaqus, the replacement of the model is easier and less time-consuming compared to Sysweld.

4 CHAPTER

FSW SIMULATION ON SYSWELD

4.1 Model building and mesh

4.1.1 Import the tool

Open **Virtual-Environment** software.
Choose **Welding and Assembly**, click on **Process Executive**, **Mesh**, **Assembly**, **Weld**, **Heat Treatment**, **Viewer**
Click **OK**

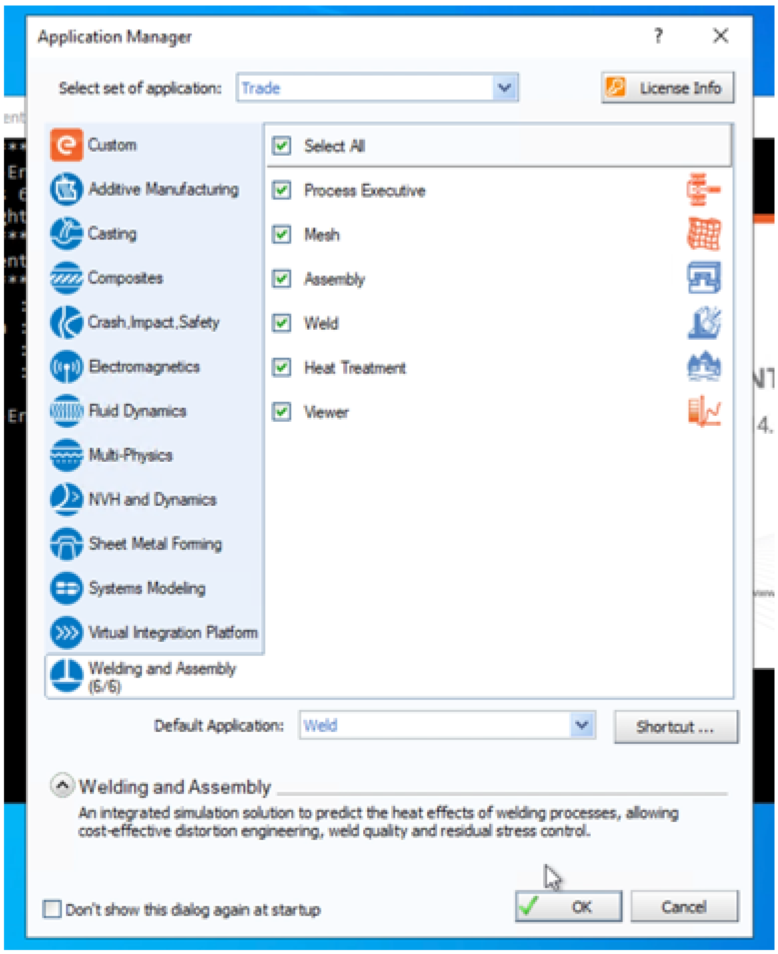


Figure 4.1.1 Application Manager UI

In **Applications** screen, click on **Mesh**.

In **Visual-Mesh** environment, click on **Open file**

Choose your cutting **Tool** obtained from any kind of CAD drawing software with your own specifications. Remeber to save your drawing in **.IGS** format

(The rotation axis of your tool should be y axis)

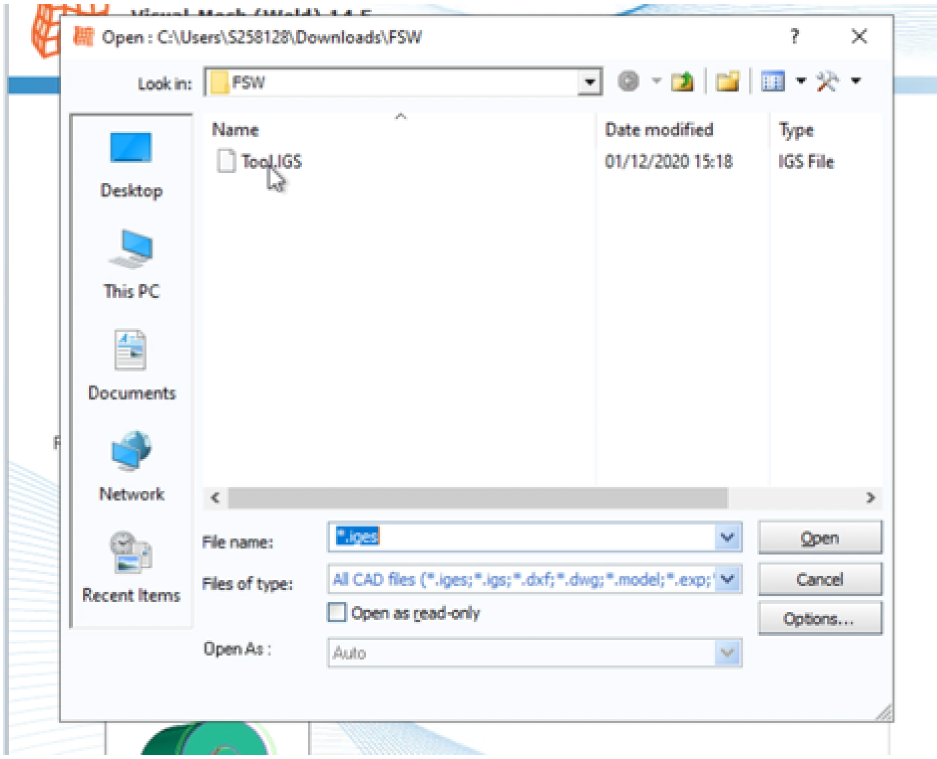


Figure 4.1.2 Import Tool

Click **Open**

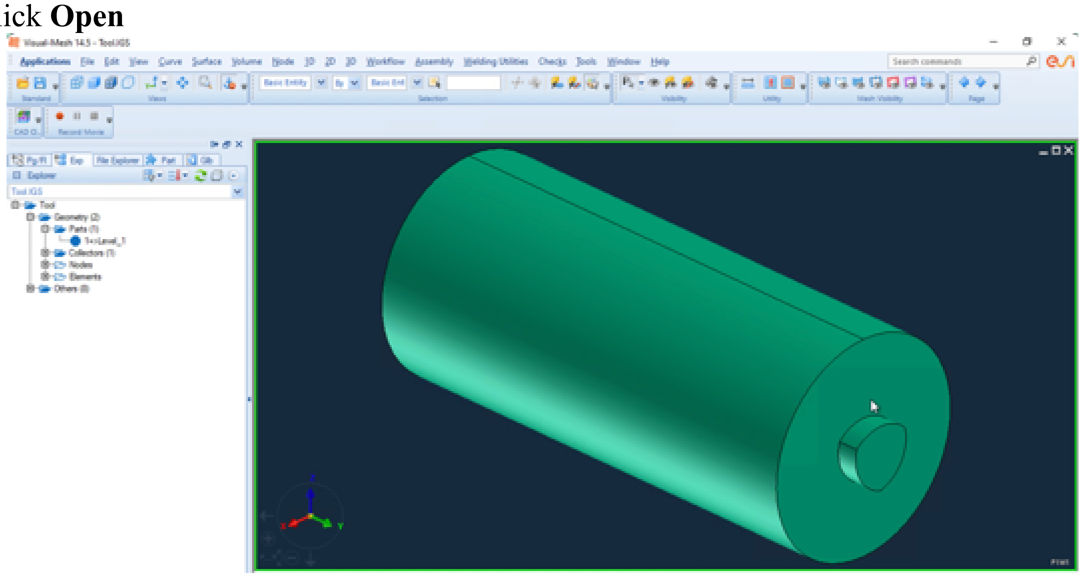


Figure 4.1.3 Tool image

4.1.2 **Draw main parts**

Draw PIN

1. Choose **Curve**, click **Circle/Arc**

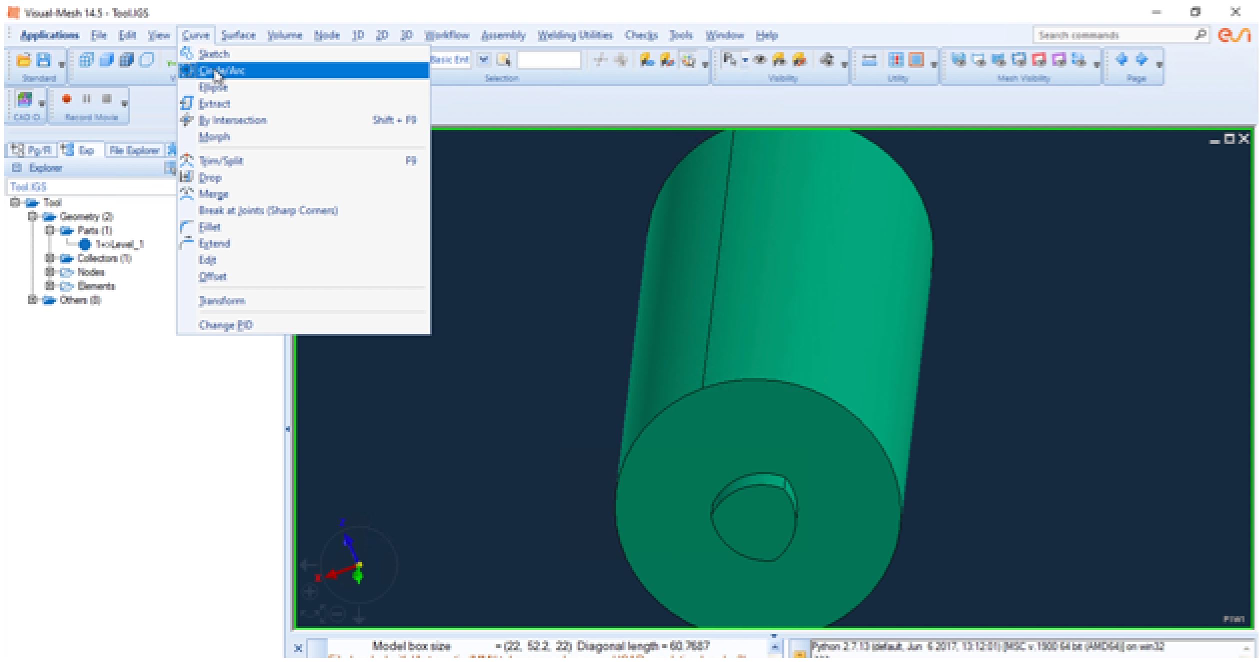


Figure 4.1.4 Curve

2. In **Circle/Arc** Window, Click **Centre-Axis**

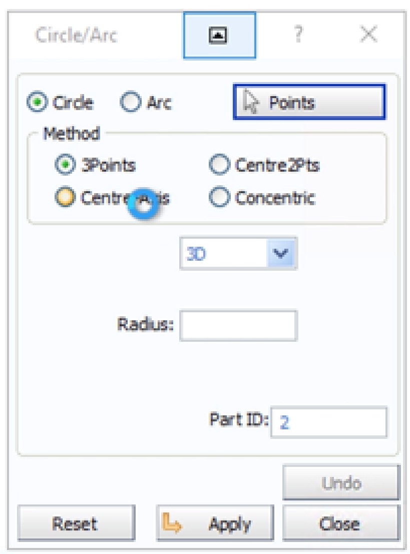


Figure 4.1.5 Circle/Arc

3. Click **Standard Axis**, along **Y Axis** and **Base Pt 0 0 0**, click OK

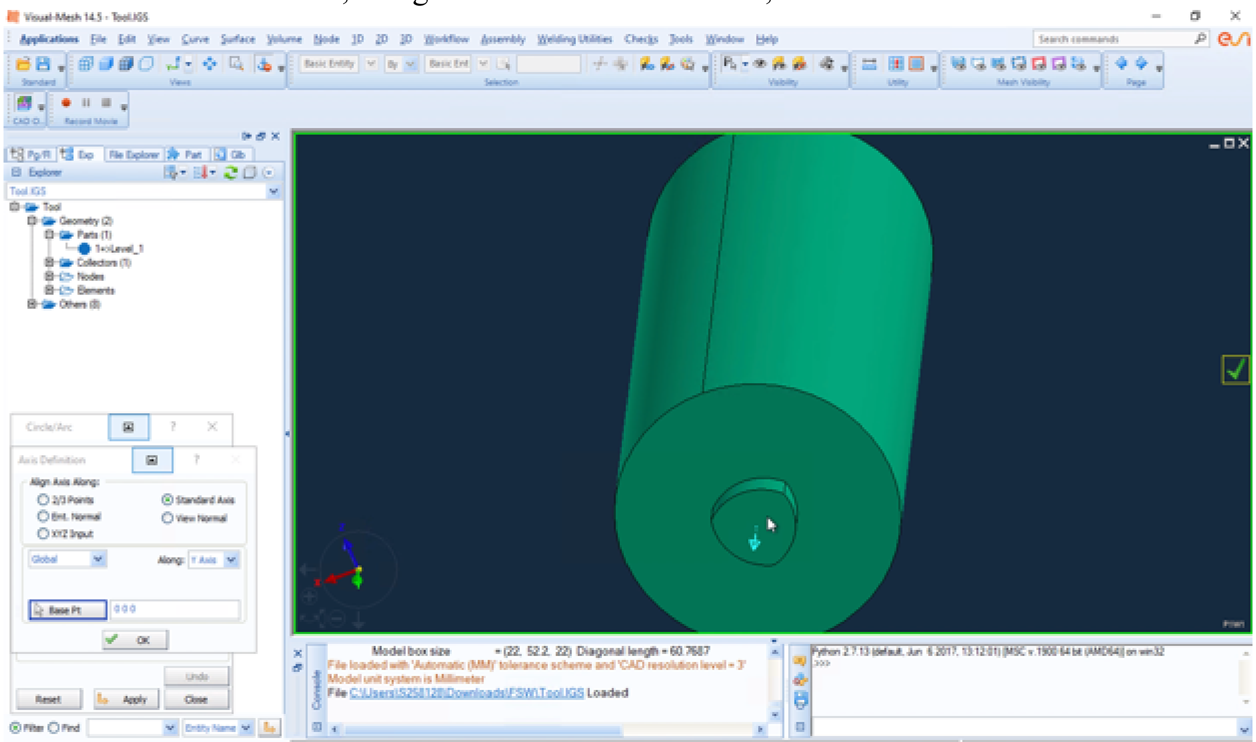


Figure 4.1.6 Axis Definition

4. In Circle/Arc window, input **Radius 4.2**, click **Apply**

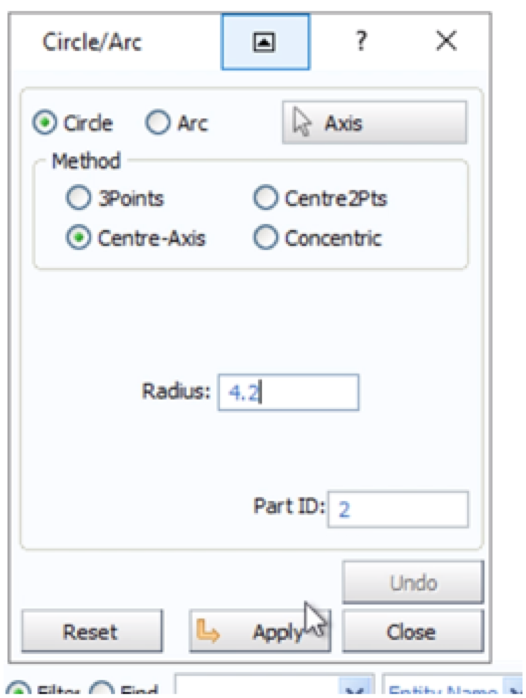


Figure 4.1.7 Radius

5. In Axis Definition window, input **Base Pt 0 2.6 0**, click **OK**
6. In Circle/Arc window, input **Radius 3.7**, click **Apply**
7. Choose **Surface**, click **Blend (Spline)**

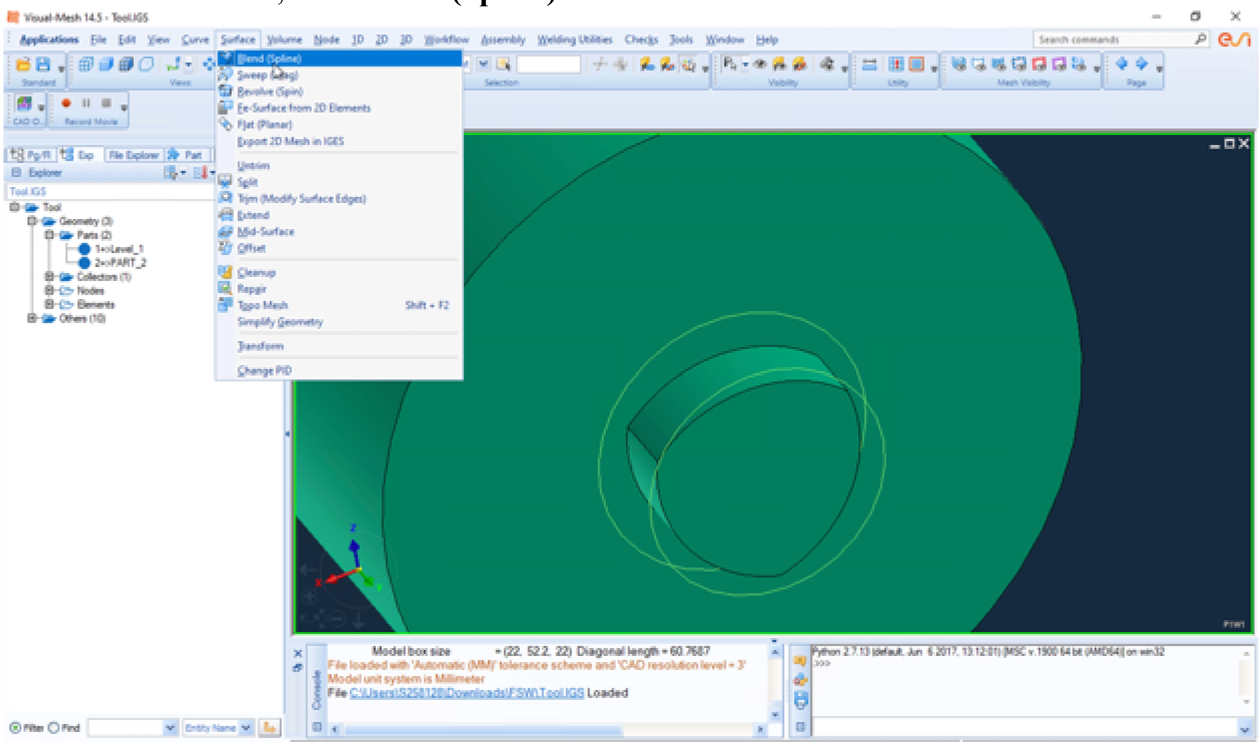


Figure 4.1.8 Blend

8. Choose these two **new curves**, click your scroll wheel to confirm your choice
9. Input **Part 3**, click **Apply**

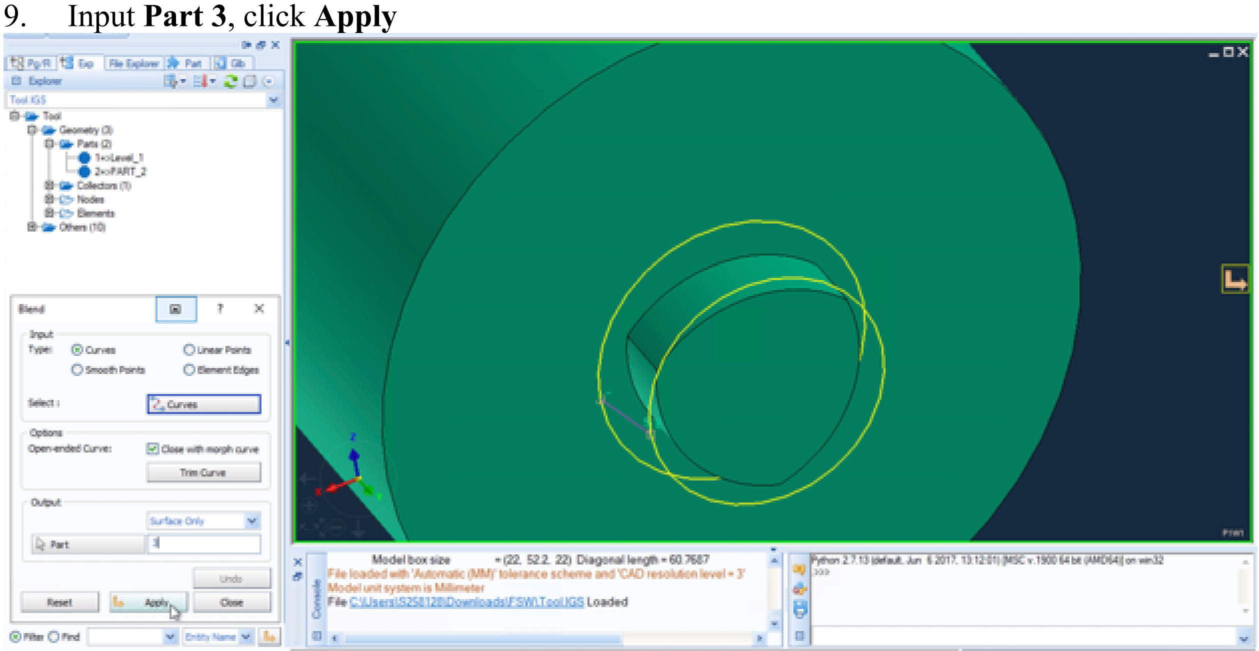


Figure 4.1.9 Part 3

10. Choose **Surface**, click **Flat (Planar)**
11. Choose the outer curve, input **Part 3** and click **Apply**

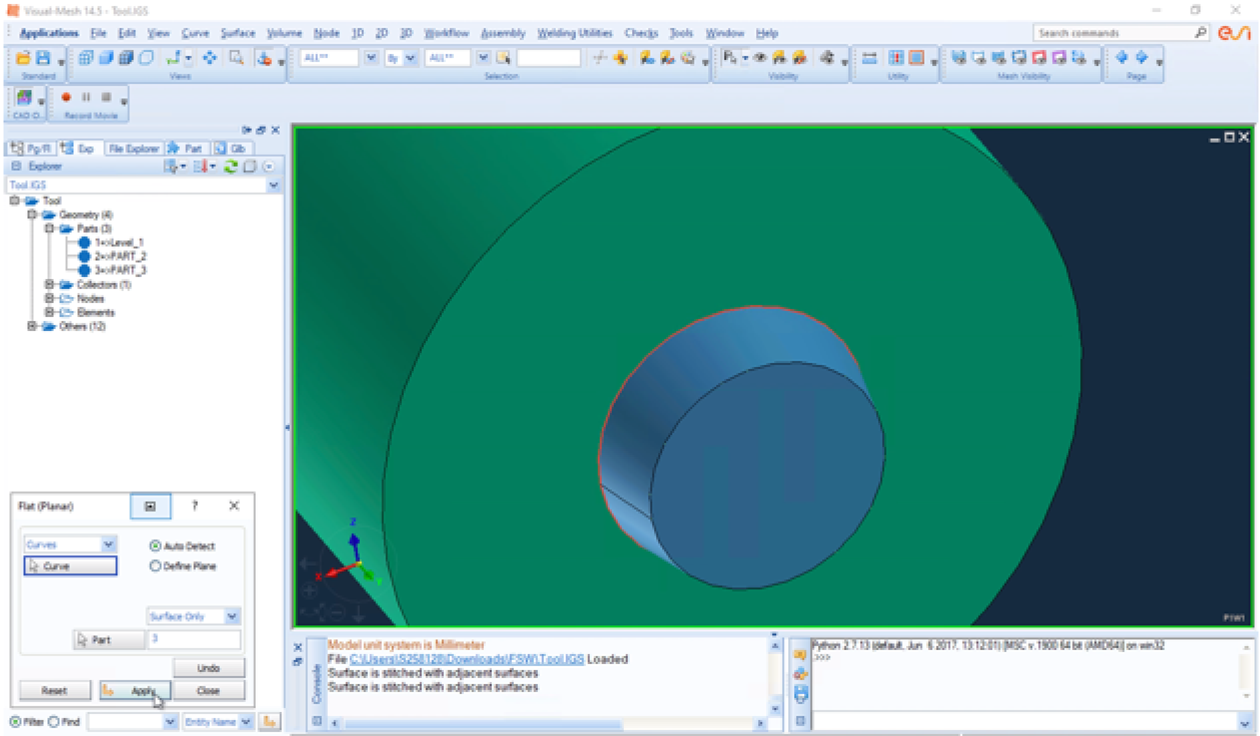


Figure 4.1.10 Part 3 with cap

Draw FLUID

1. Choose **Curve**, click **Circle/Arc**
2. In **Circle/Arc** window, input **Radius 12.5**
3. Click **Apply**
4. In **Axis Definition** window, input **Base Pt 0 2.9 0**, click **OK**
5. Click **Apply**

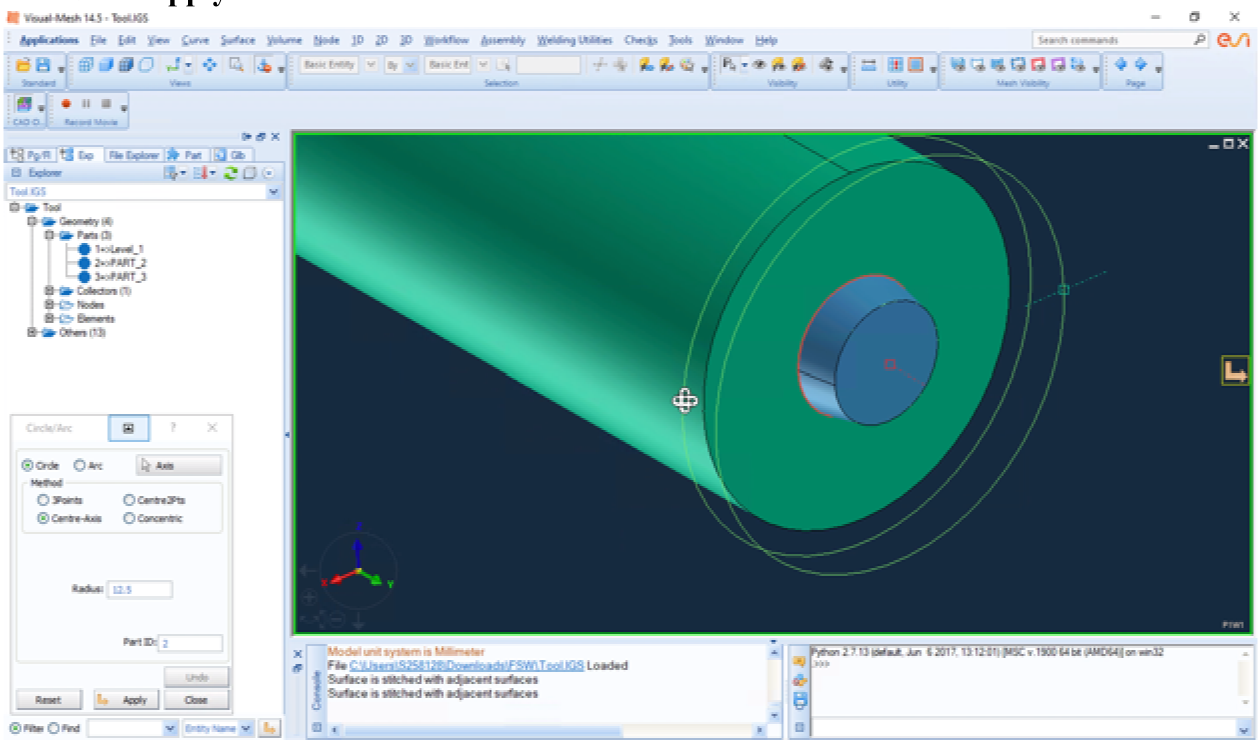


Figure 4.1.11 Fluid 2 curves

6. Choose **Surface**, click **Blend (Spline)**
7. Choose these 2 new drawn curves, input **Part 4**, click **Apply**
8. Choose **Surface**, click **Flat (Planar)**
9. Choose the outer curve, input **Part 4** and click **Apply**

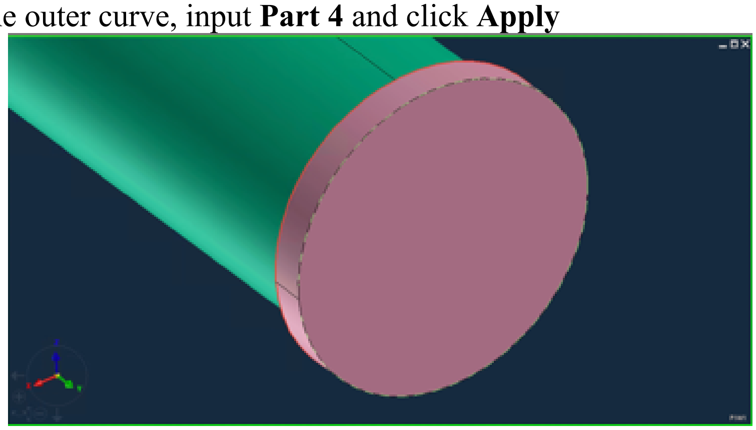


Figure 4.1.12 Fluid

10. Choose **Surface**, click **Flat (Planar)**
11. Choose tool cylinder outer curve and Fluid upper curve

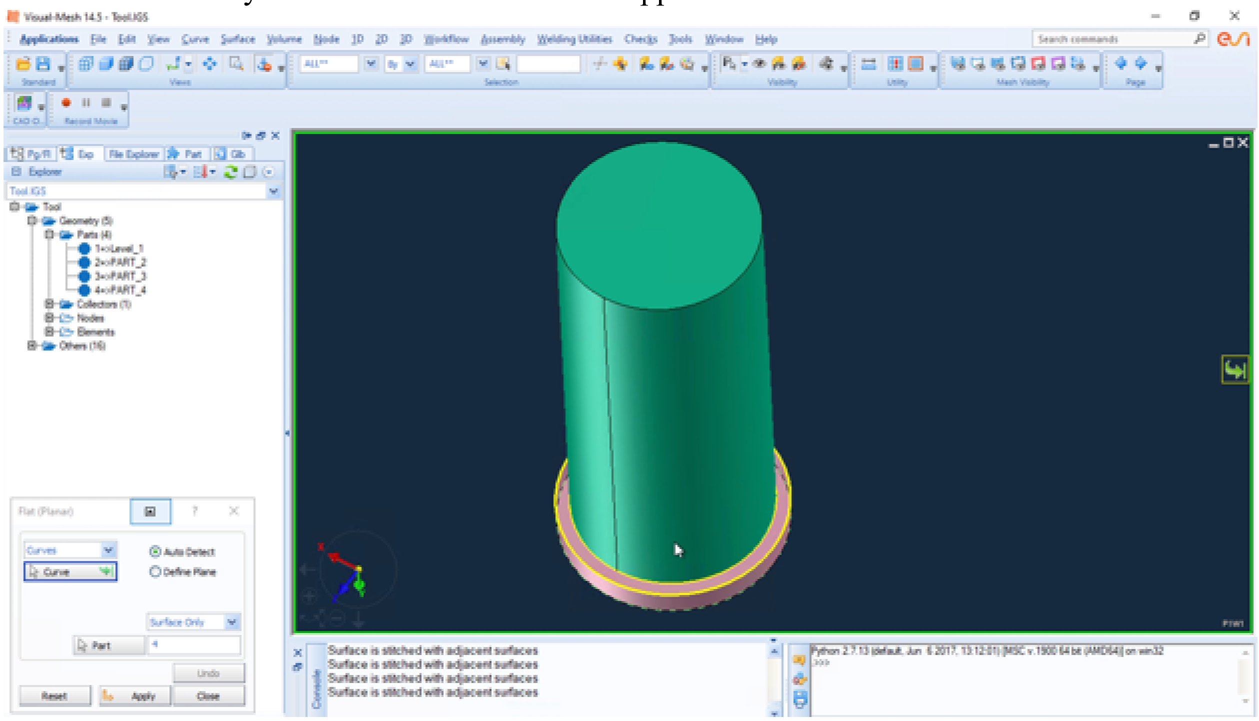


Figure 4.1.13 Two curves

12. Click **Apply**

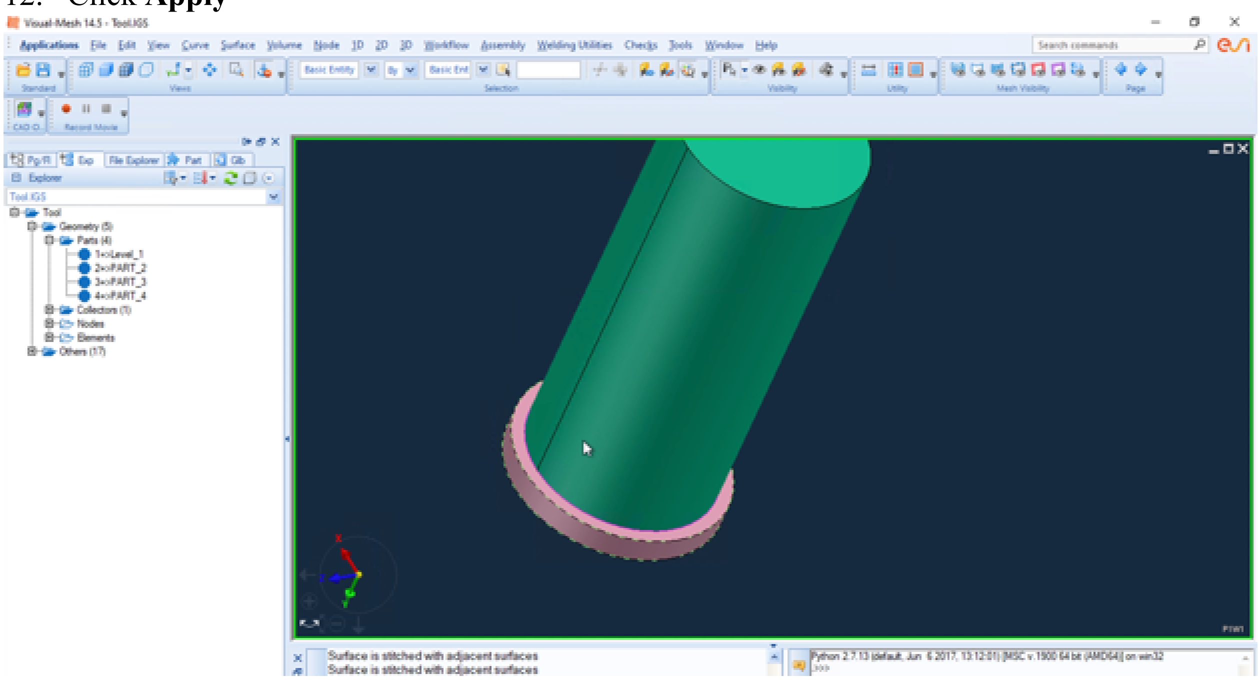


Figure 4.1.14 FLUID

Rename Part

1. Choose from **Part list**, right click on **1=>Level_1**, choose **Part Manager**

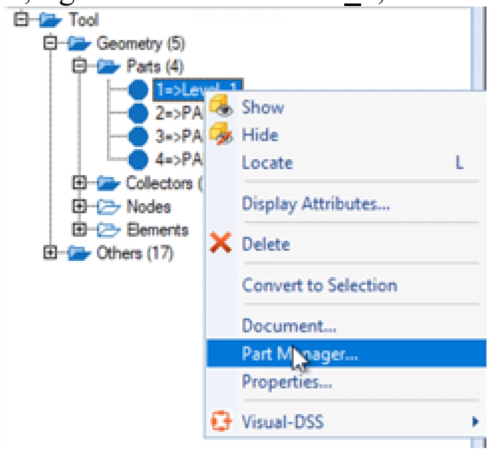


Figure 4.1.15 Part Manager

2. In **Part Manager** window, Input name **TOOL**, click **Close**

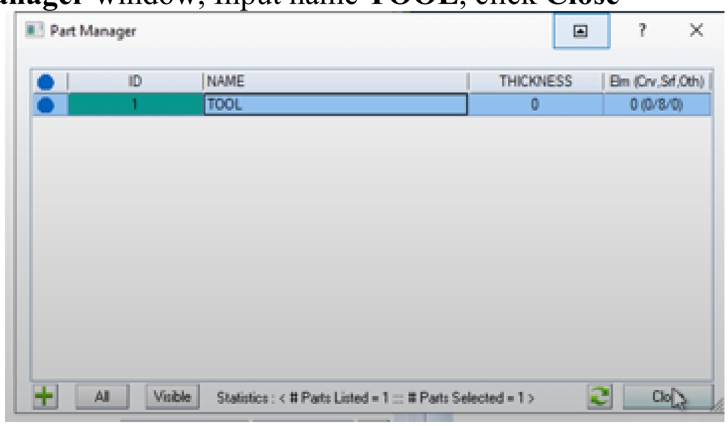


Figure 4.1.16 Rename

3. With the same method, rename **Part_2** as **CURVES**
4. Rename **Part_3** as **PIN**
5. Rename **Part_3** as **FLUID**

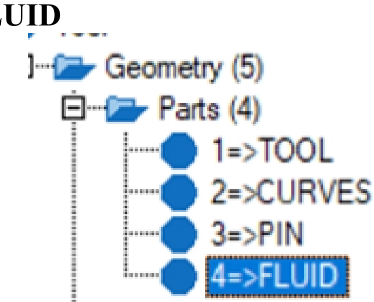


Figure 4.1.17 Part list

Draw SHEET

1. **Hide** parts **TOOL**, **CURVES** and **PIN**
2. Choose **Volume**, click **Box**

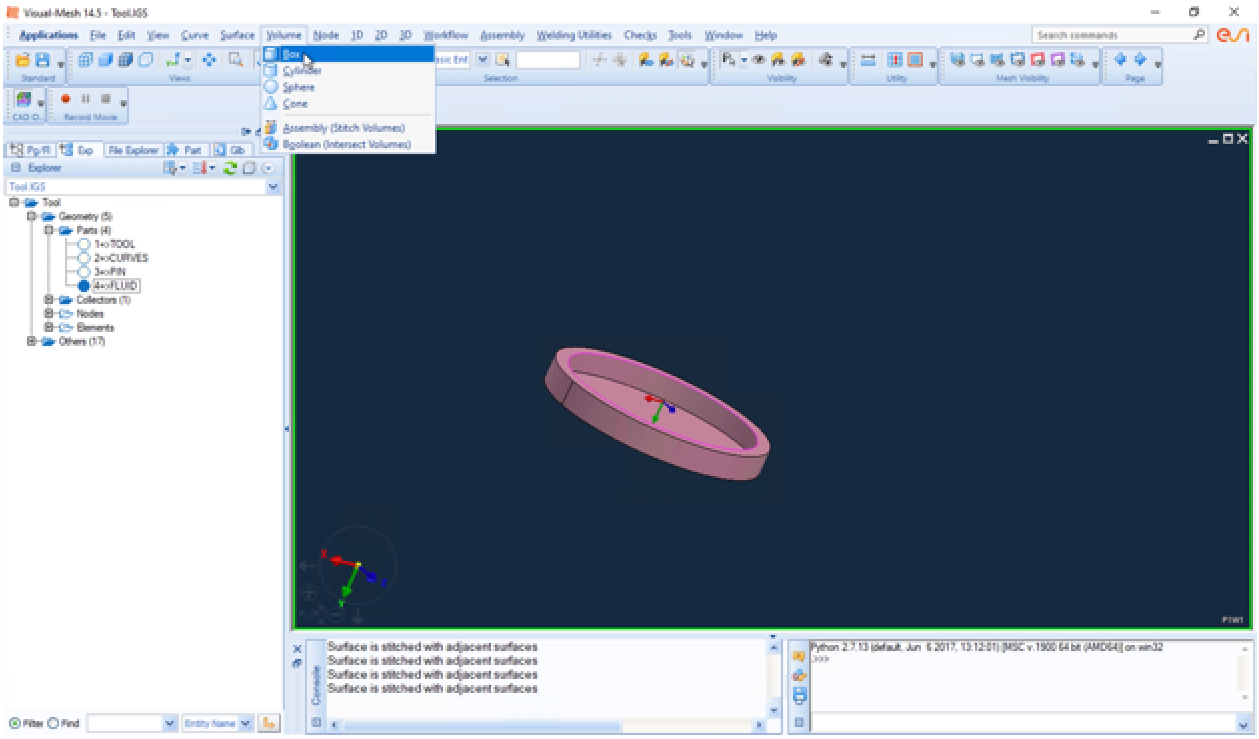


Figure 4.1.18 Fluid

3. In **Box** window, input **Width 3.5**, **Height 200** and **Depth 300**, **Part 5** and click **Apply**

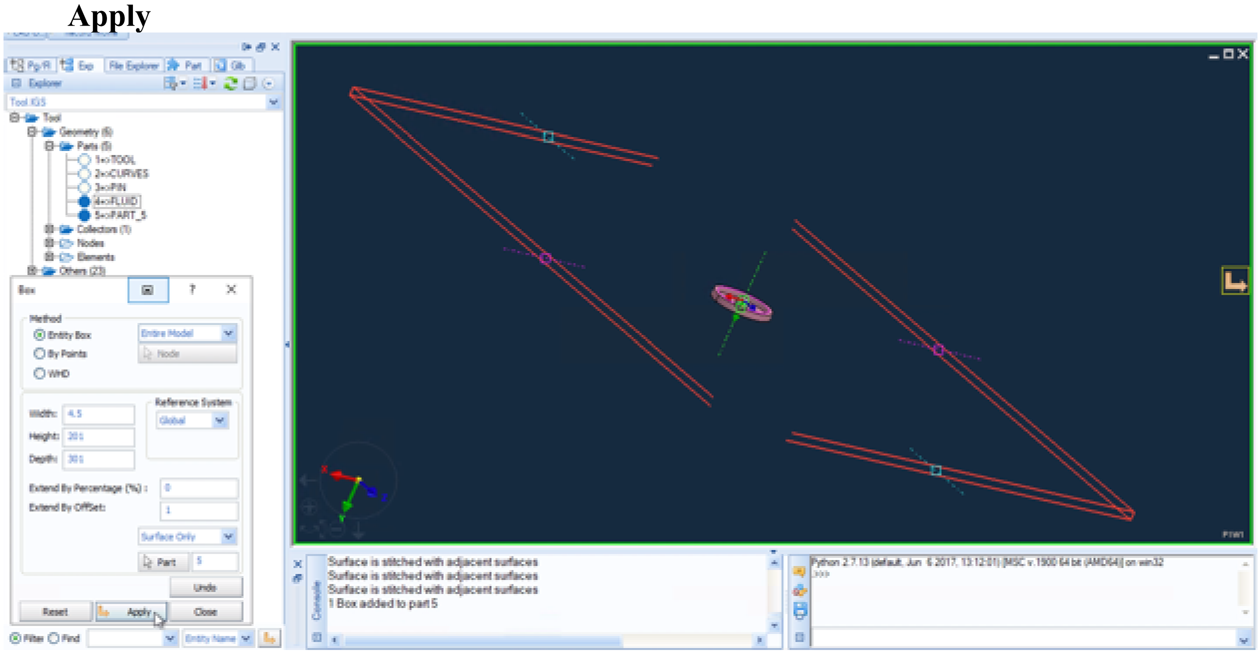


Figure 4.1.19 Box

4. Choose **Tools**, click **Transform**

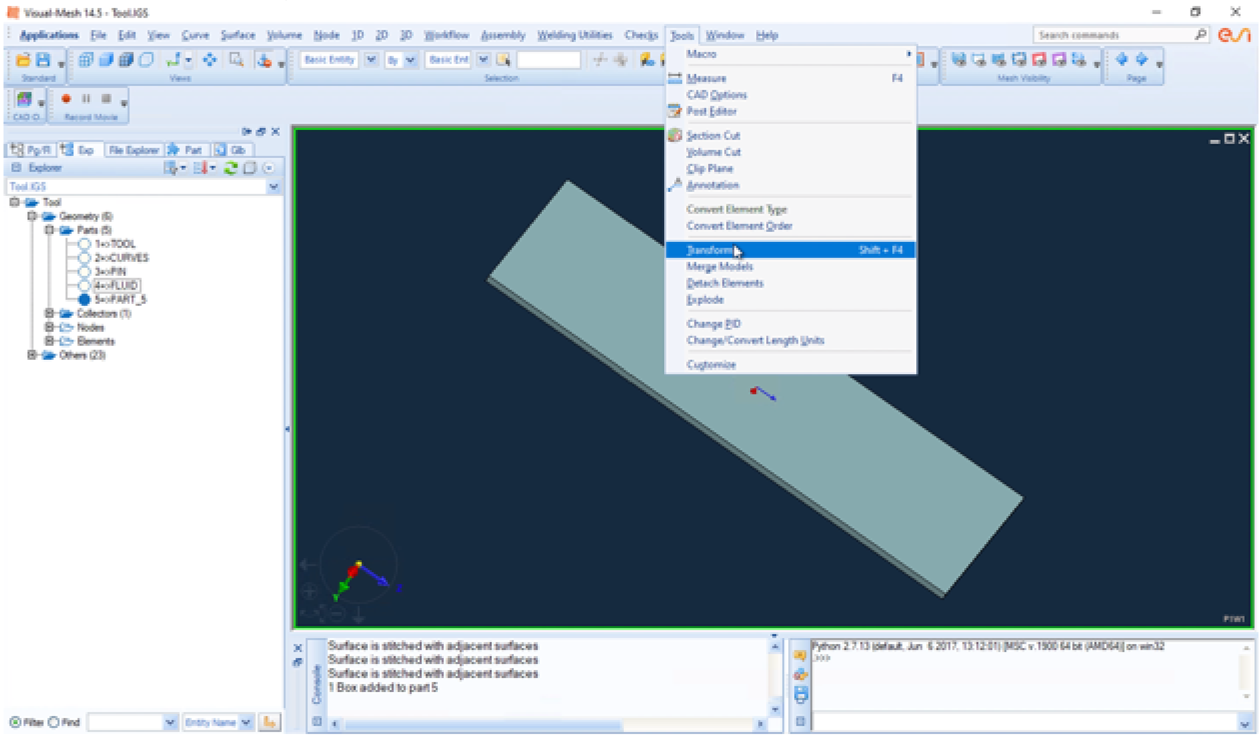


Figure 4.1.20 Transform

5. In **Transform** window, choose **Surface** and include all **Part 5**
6. Input **dY 0.3**, click **Move**

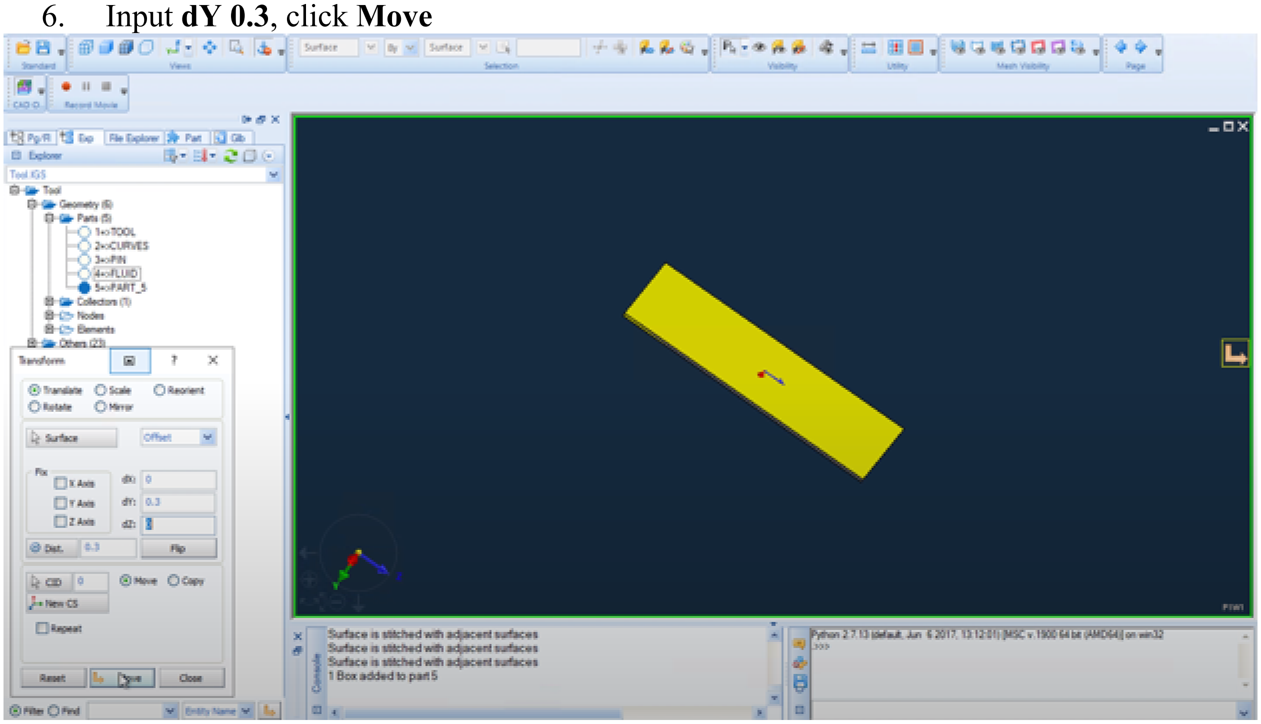


Figure 4.1.21 Transform

Draw SUPPORT

1. Hide **FLUID**
2. Choose **Volume**, click **Box**
3. In **Box** window, input **Width 150**, **Height 200** and **Depth 300**, **Part 6** and click **Apply**
4. Hide **Part 5**
5. Choose **Tools**, click **Transform**
6. In **Transform** window, choose **Surface** and include all **Part 6**
7. Input **dY 76.75**, click **Move**

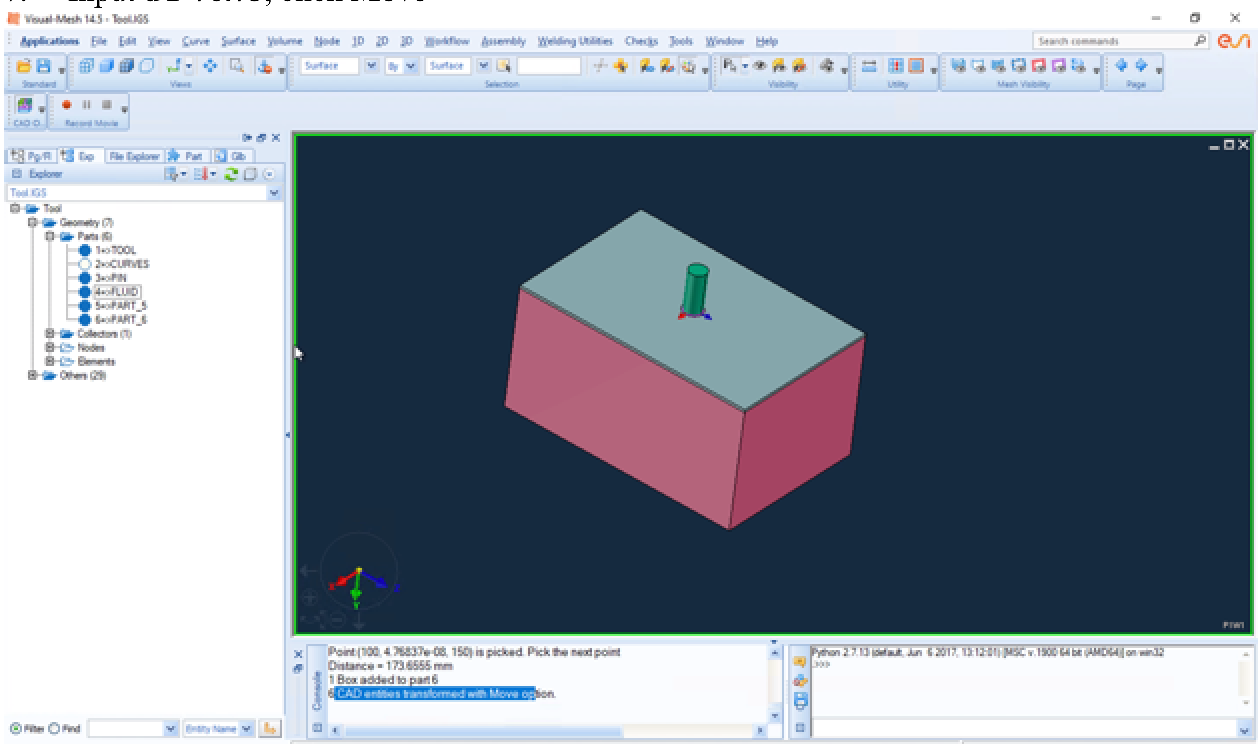


Figure 4.1.22 Entire model

Dealing with Details

1. Only show **TOOL** and **Part 5**
2. Choose **Surface**, click **Split**

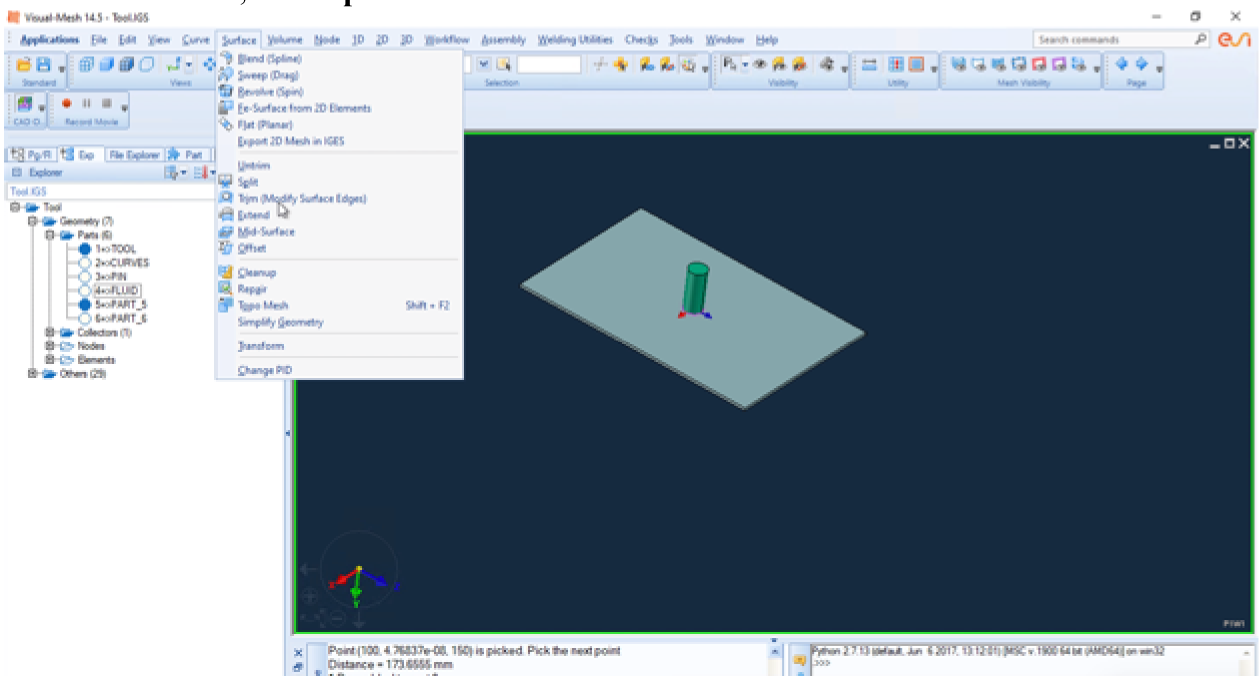


Figure 4.1.23 Split

3. Include **top surface of Part 5**, split by **Plane**

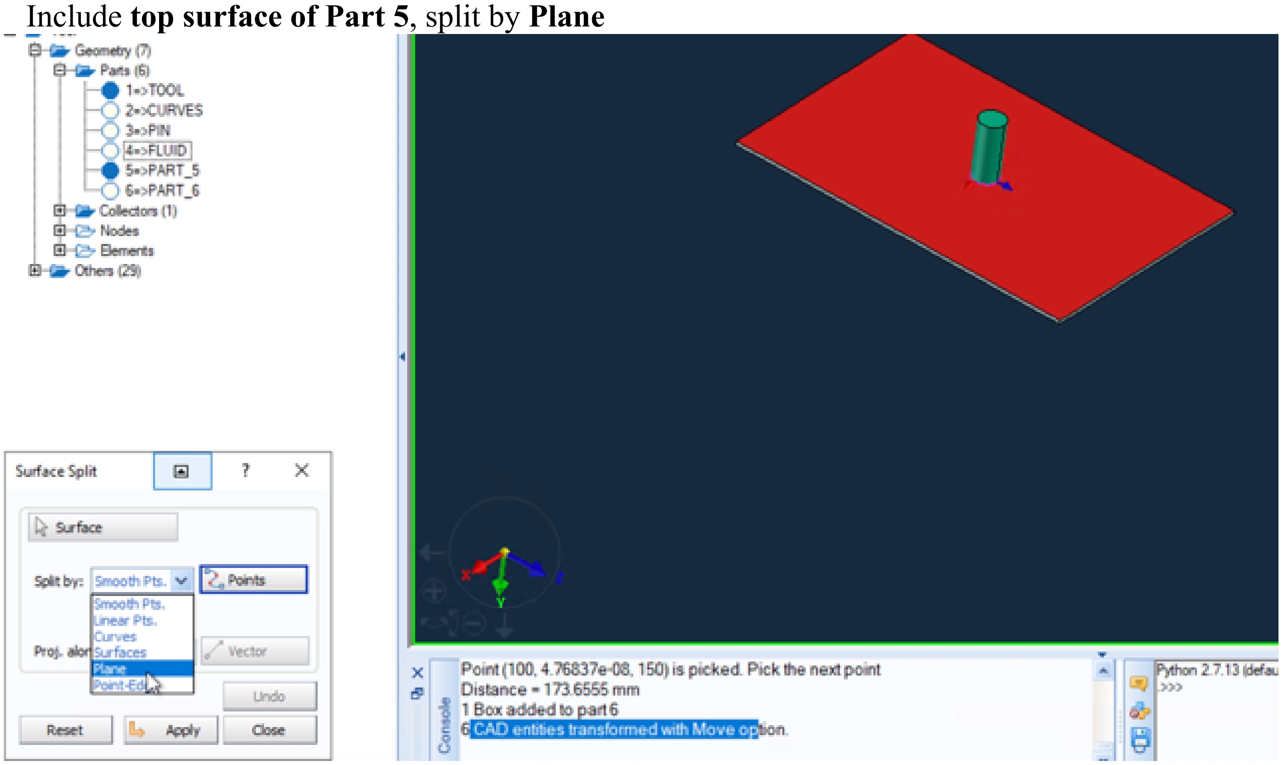


Figure 4.1.24 Surface Split

4. In **Plane Definition** window, Along **Z Axis**, input **Base Pt 0 0 0**, click **OK**, click Apply

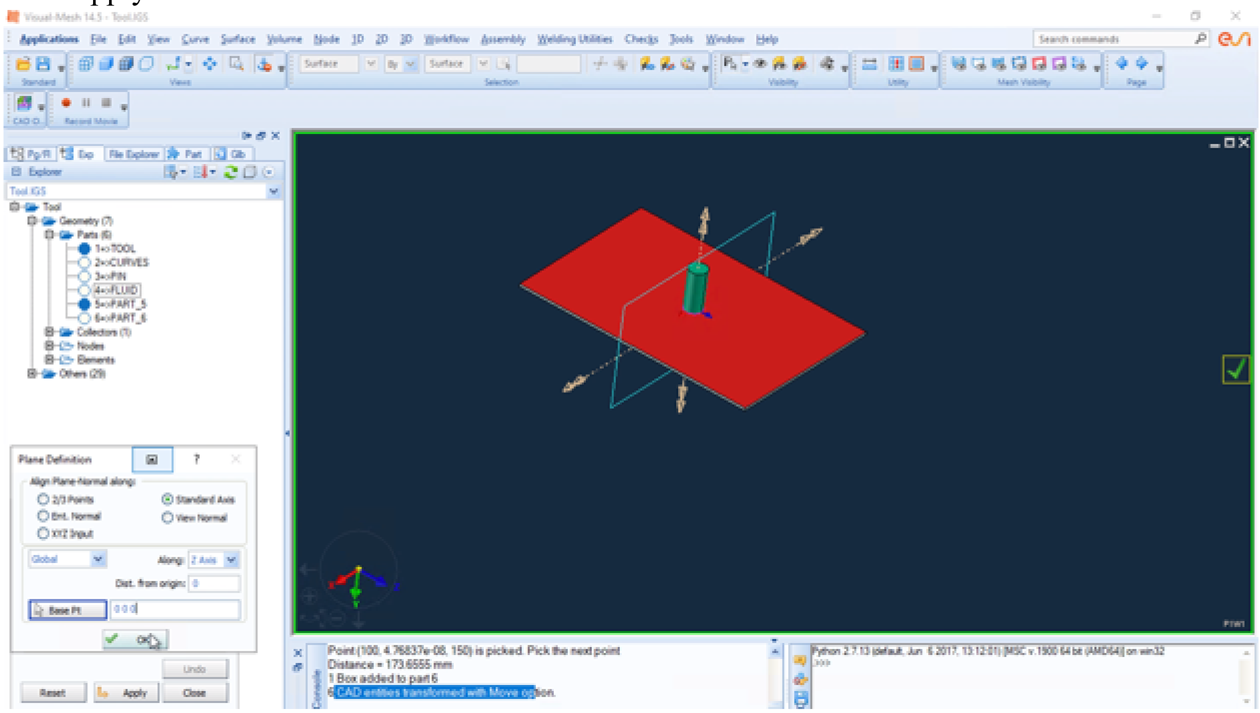


Figure 4.1.25 Surface Split of sheet

5. Show **TOOL**, **CURVES** and **PIN**
6. Choose the **bottom surface** of the tool and split by the pin's outer curve

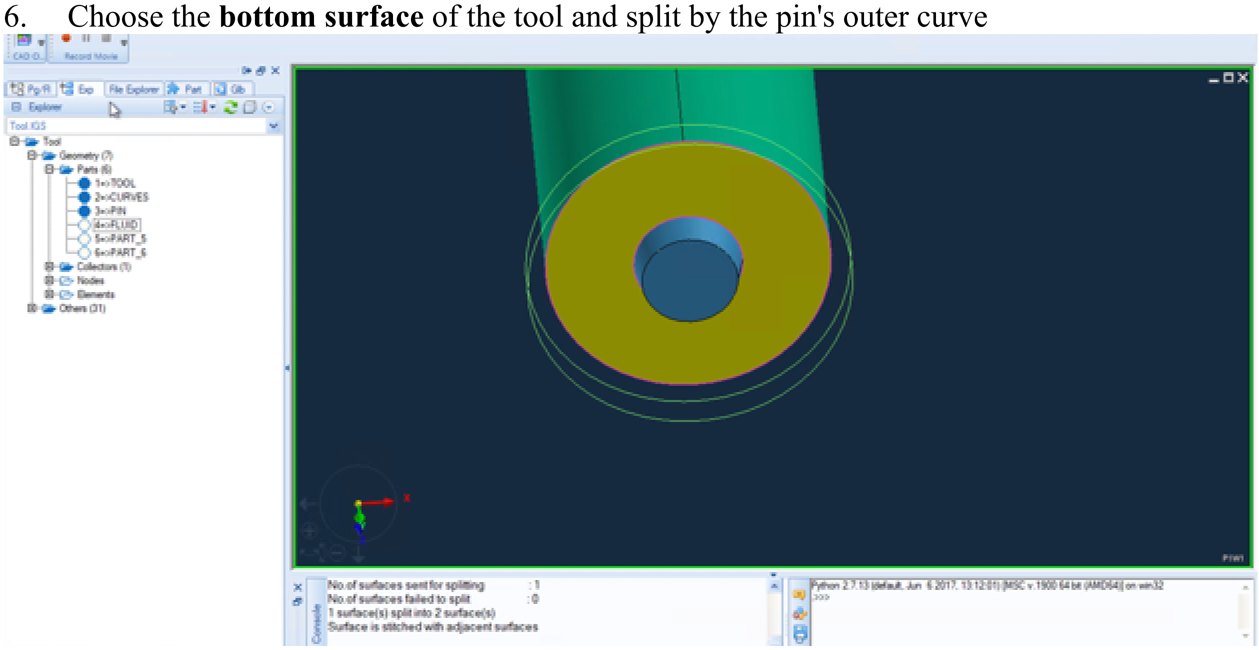


Figure 4.1.26 Surface Split of the tool

7. Rename **Part 5** as **SHEET**
8. Rename **Part 6** as **SUPPORT**

4.1.3 Mesh and Collectors

Mesh the Tool

1. Show only the **TOOL** part
2. Choose **2D**, click **Automesh Surfaces**

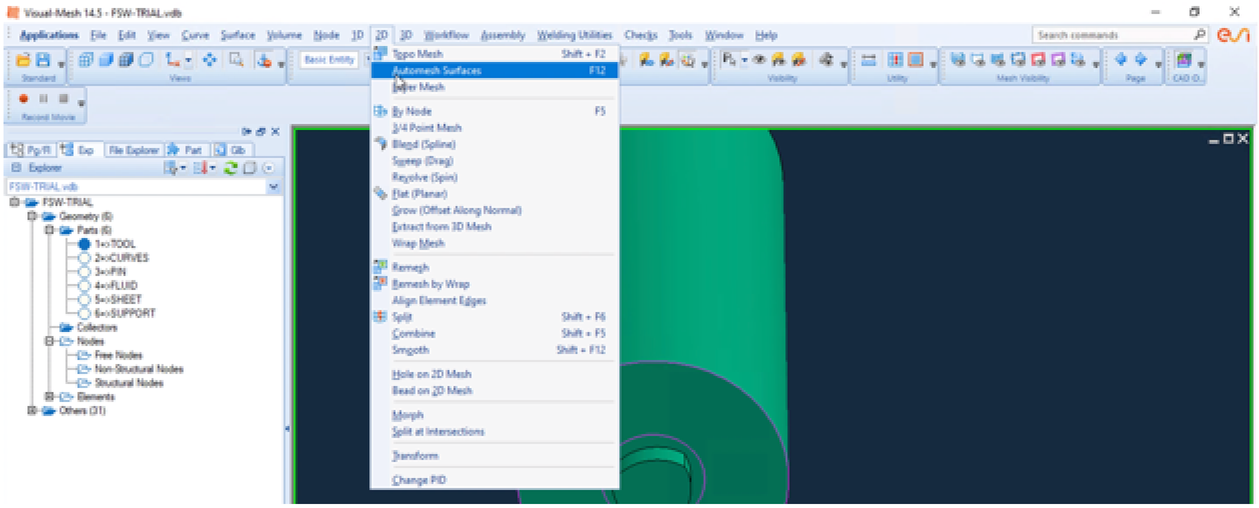


Figure 4.1.27 Automesh Surfaces

3. Choose the **bottom surface of the tool**, **Set Element Size 0.3,**
4. In Method window, choose **Type Tria**. In **ID** window, input **Part 7**

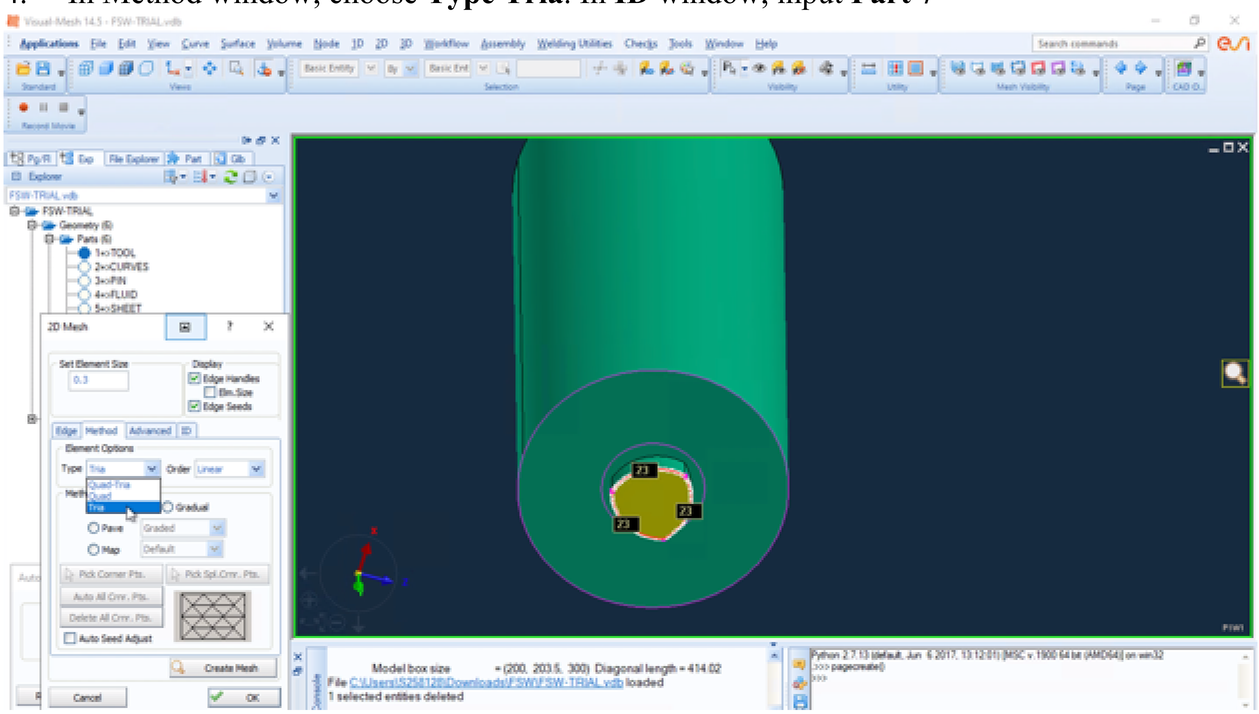


Figure 4.1.28 2D Mesh

5. Click **Create Mesh**. Click **OK**

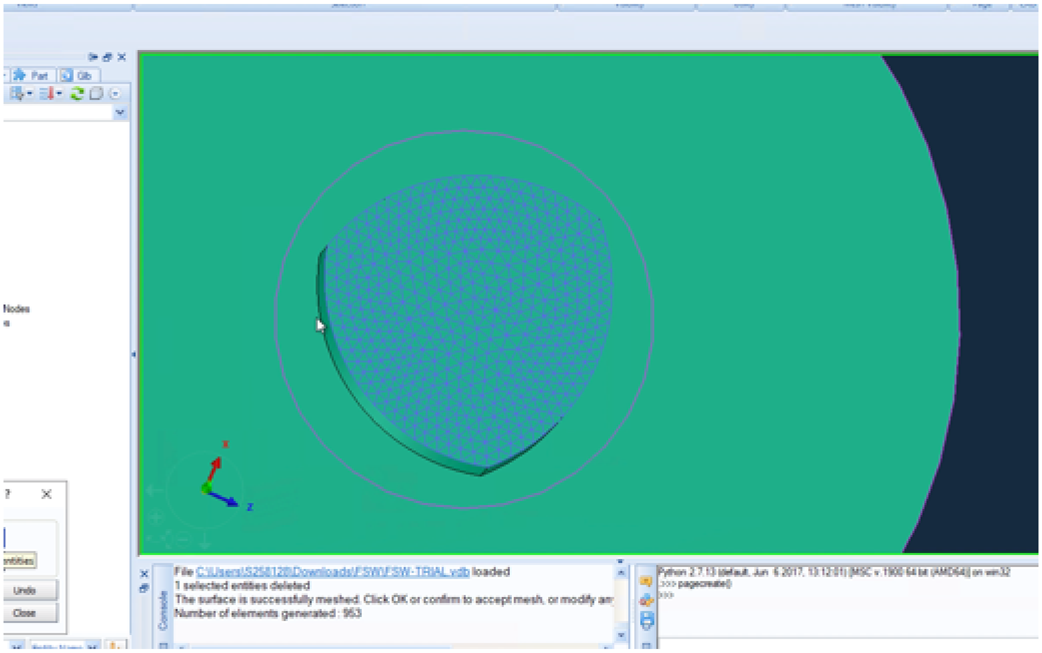


Figure 4.1.29 Mesh bottom surface

6. Select **surrounding surfaces** of the cutter head.
7. In **Advanced** window, click **Within Part** and **Across Part**

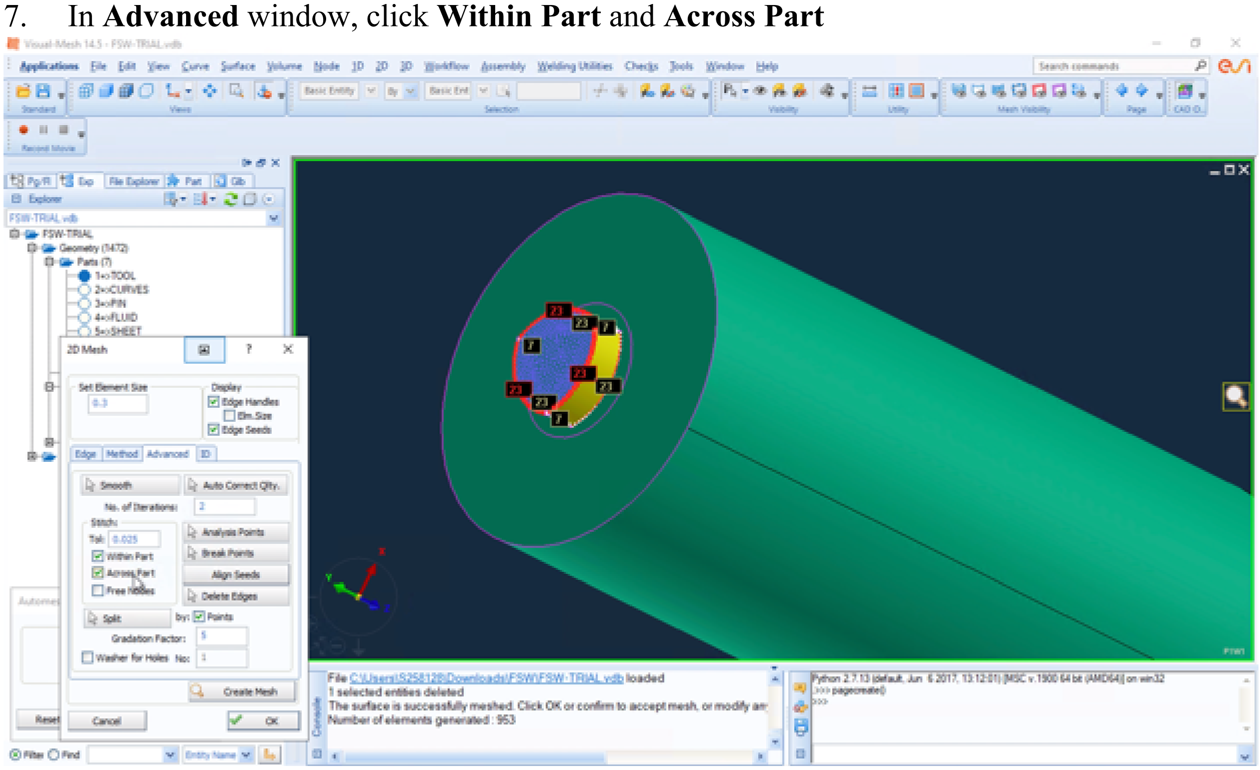


Figure 4.1.30 Mesh bottom surface

8. In Method window, choose **Pave** and **Graded**
9. Click **Create Mesh**, click **OK**

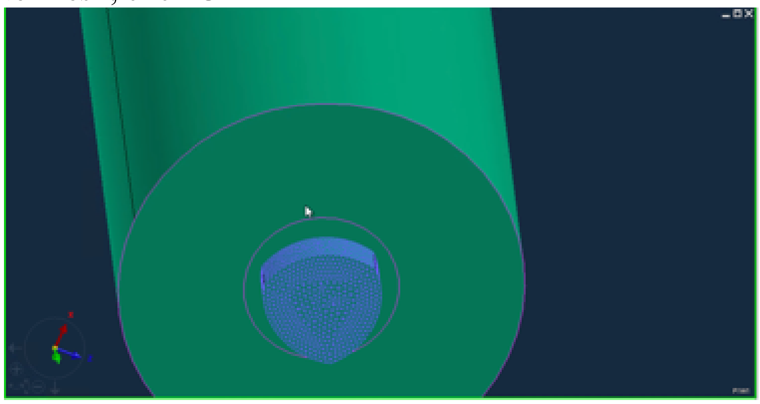


Figure 4.1.31 Mesh

10. Select the adjacent surface of smaller inner circle
11. **Set Element Size 0.4**, in **Method** Window, choose **Auto**
12. Click **OK**

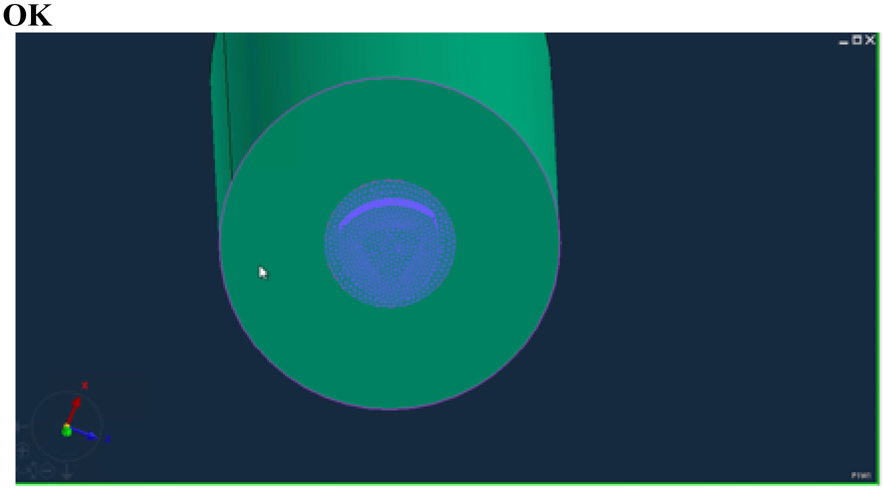


Figure 4.1.32 Mesh

13. Select the adjacent surface.
14. **Set Element Size 0.7**, click **OK**

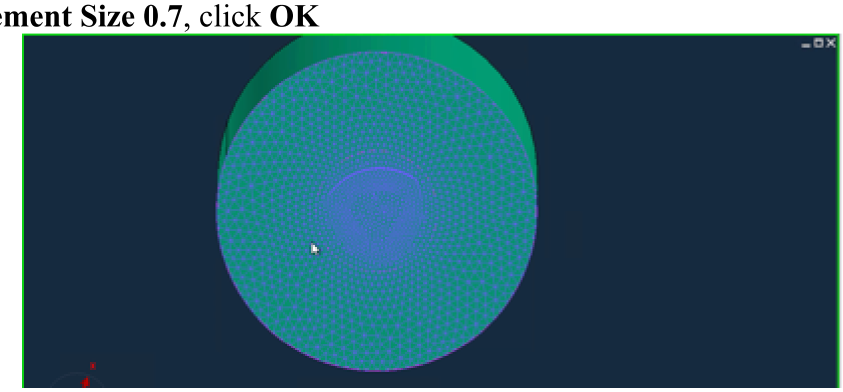


Figure 4.1.33 Mesh

15. Select two adjacent surfaces
16. **Set Element Size 1**
17. Reduce the number of edge nodes from **25 to 20**, from **35 to 19**

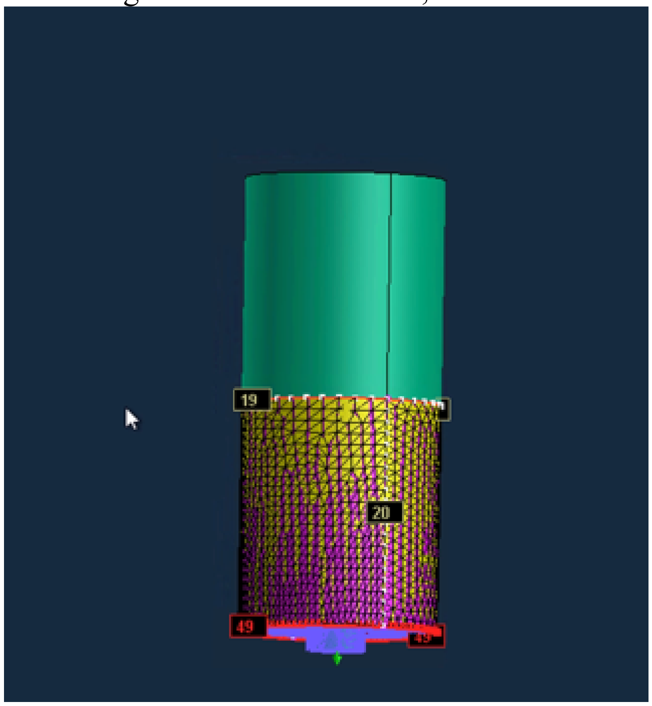


Figure 4.1.34 Mesh

18. In **Edge** window, **Biasing**: choose **Type Linear**, **Bias Edge** to bias the edge with 20 nodes
19. Click **OK**

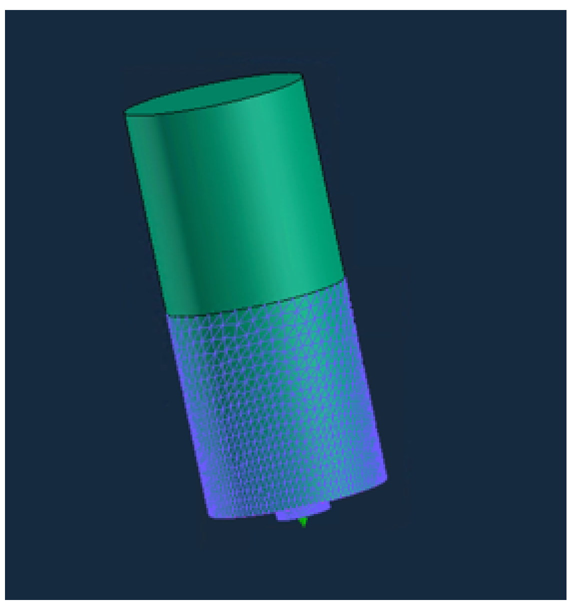


Figure 4.1.35 Mesh

20. Select two adjacent surfaces
21. Reduce the number of edge nodes from **25 to 16**, from **35 to 13**
22. Click **OK**

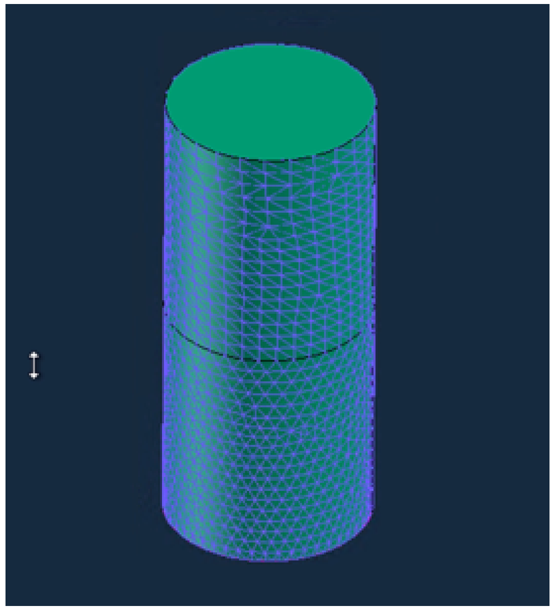


Figure 4.1.36 Mesh

23. Choose the **Top surface**
24. Click **OK**

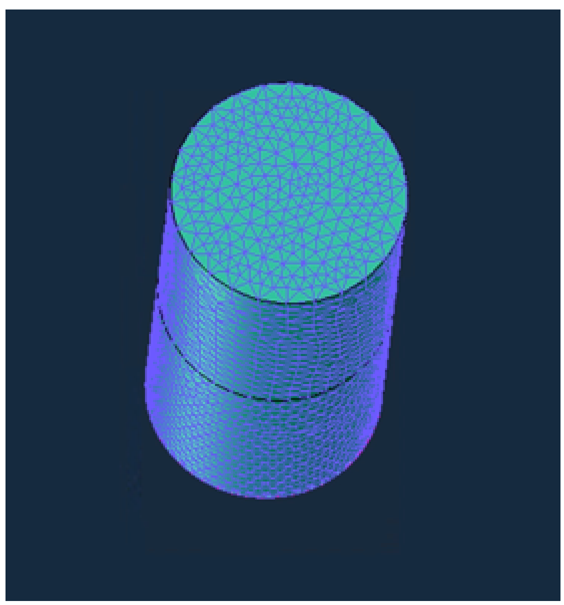


Figure 4.1.37 Mesh

25. Rename **PART_7** as **2D_TOOL**

26. Choose **3D**, click **Tetra Mesh**

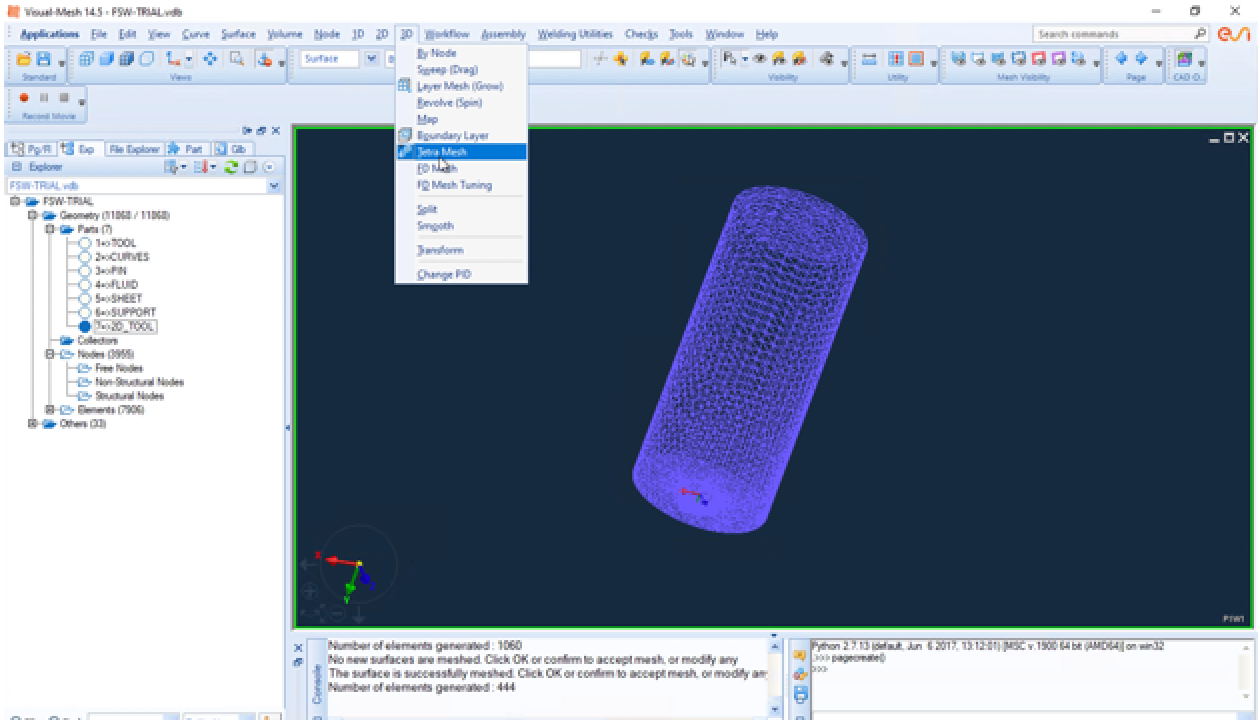


Figure 4.1.38 Tetra Mesh

27. Select all the elements in Part 7
28. In **Tetra Mesh** window, input **Transition Factor 1.3**
29. Click **Mesh**, click **Apply**

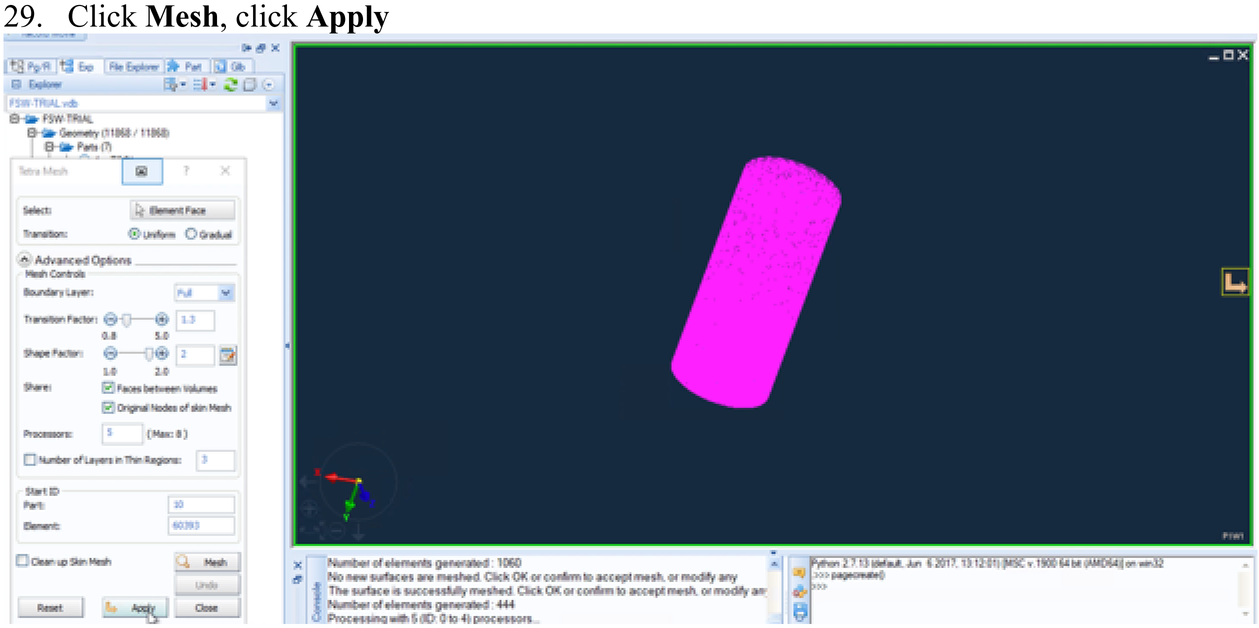


Figure 4.1.39 Tetra Mesh

30. Rename **Part_8** as **3D_TOOL**

Define Collectors 1-3

1. In **Selection** window, choose **2D Elements**, choose **Adjacent elements button**

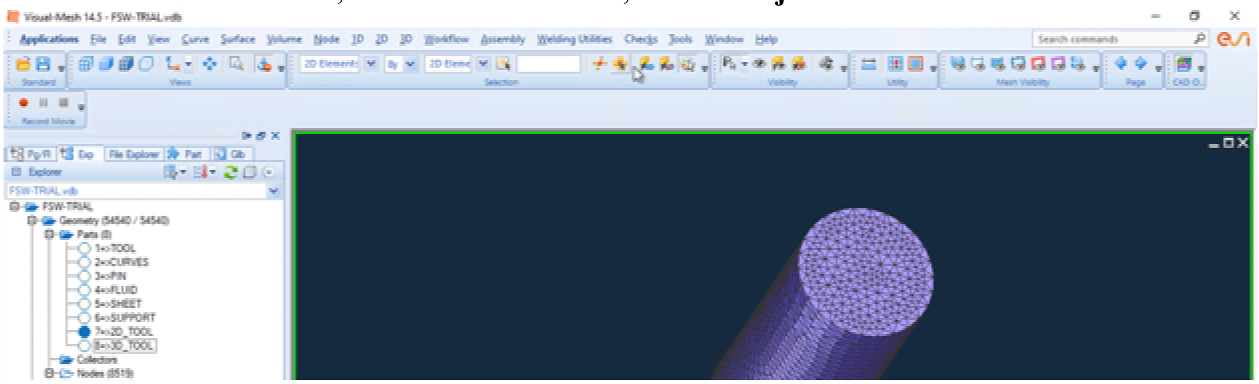


Figure 4.1.40 Adjacent elements

2. Select the **top surface**, **right click** on it **Tools**, then **Add to New Collector**

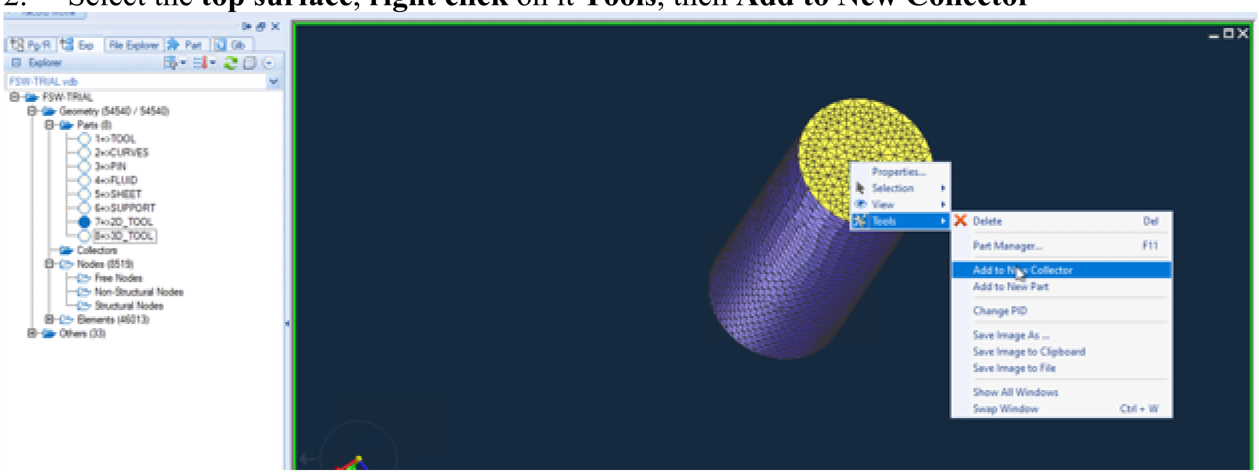


Figure 4.1.41 New Collector

3. **Right Click** on **Collector_1**, choose **Edit**

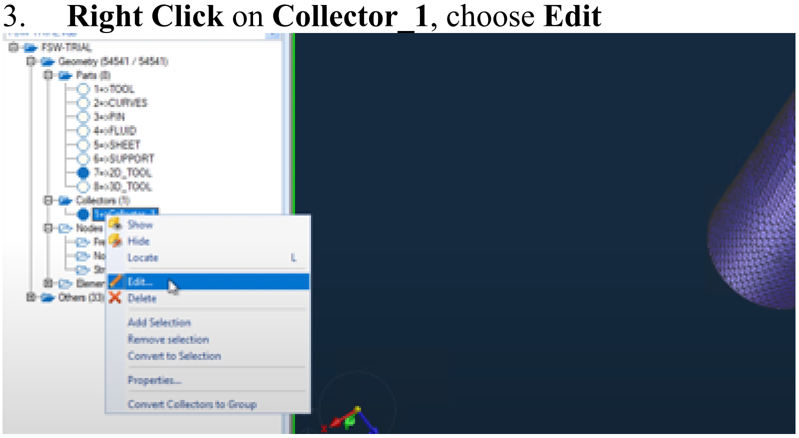


Figure 4.1.42 Edit Collector

4. Rename **Collector_1** as **TOOL_MACHINE**

5. In **Selection** window, choose **2D Elements**, choose **Adjacent elements button**
6. Select the **side surface**, **right click** on it **Tools**, then **Add to New Collector**

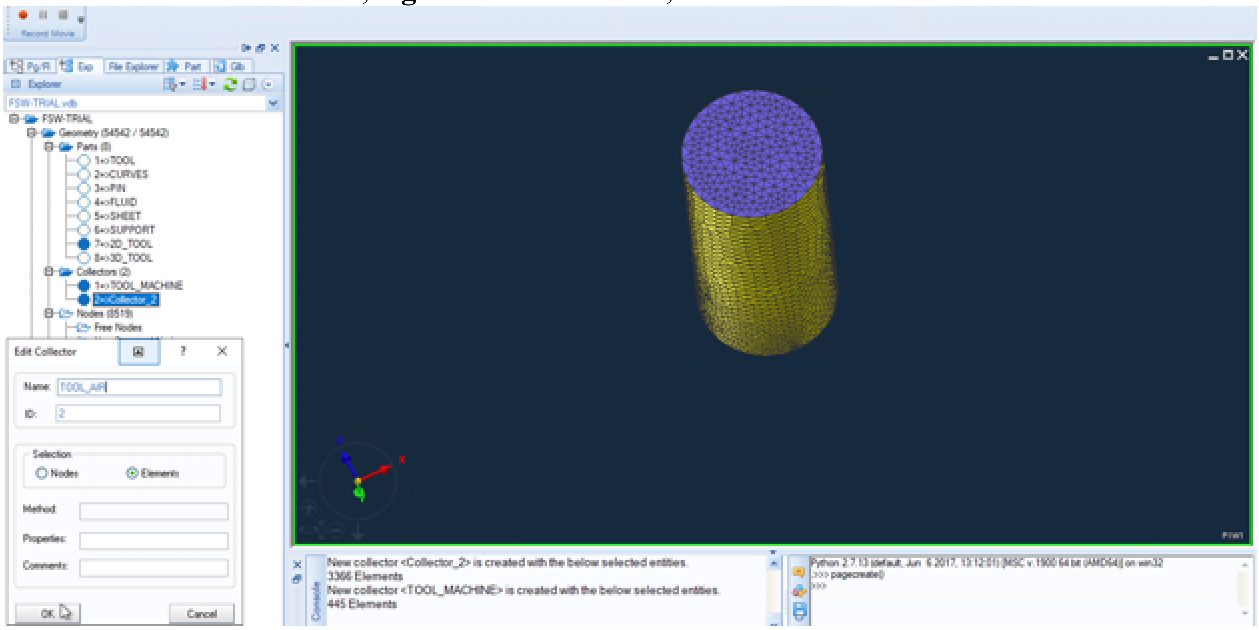


Figure 4.1.43 TOOL_AIR

7. Rename **Collector_2** as **TOOL_AIR**
8. Select **all 3D elements** of part **3D_TOOL**, then **Add to New Collector**
9. Rename **Collector_3** as **M3DO**

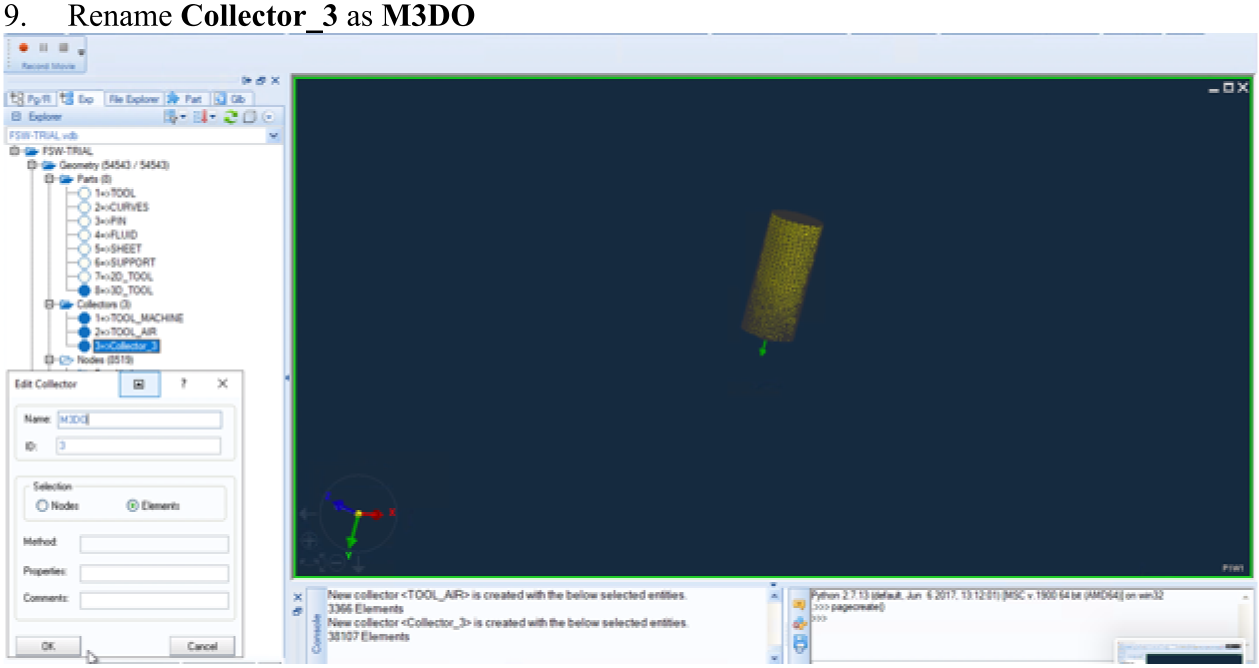


Figure 4.1.44 M3DO

MESH PIN part & Collector_4

1. Show only **part 7** and hide all the top elements

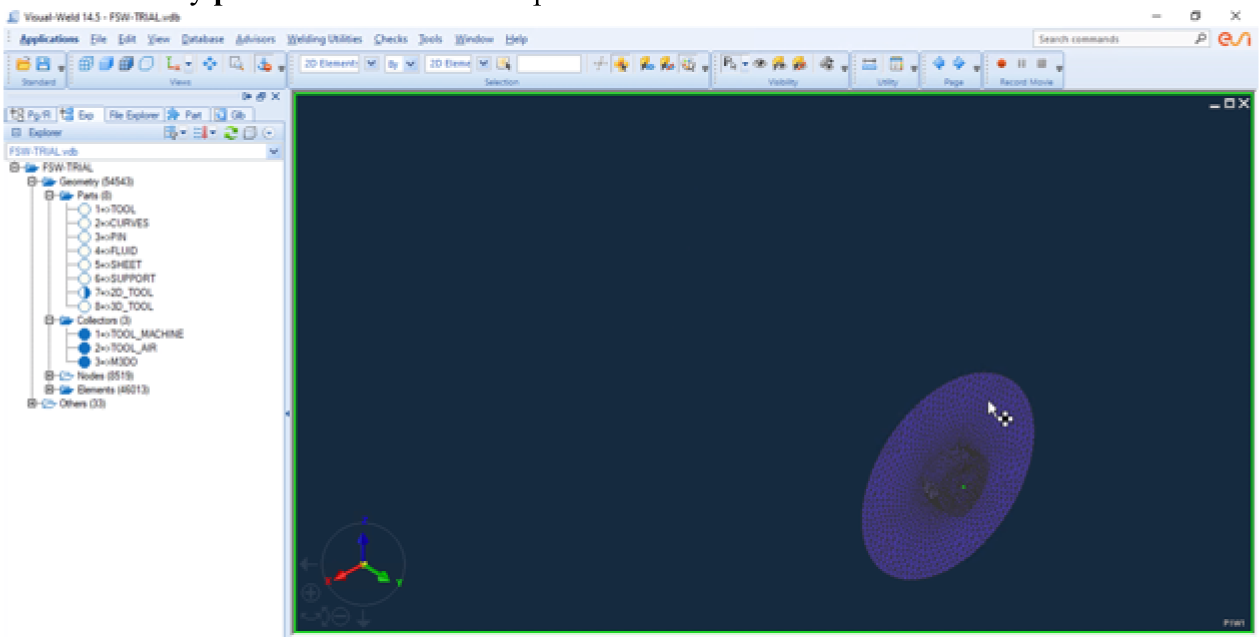


Figure 4.1.45 Part of 2D_TOOL

2. In **Views** window, choose **Right-Side View**

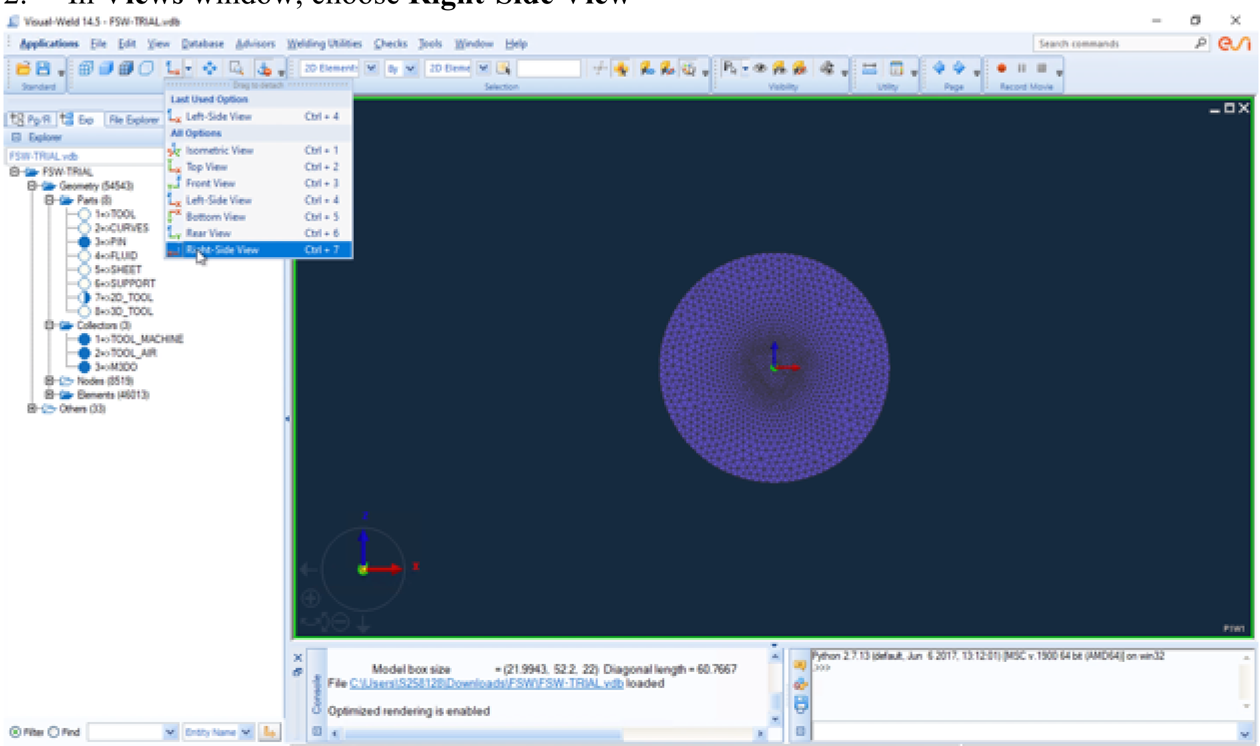


Figure 4.1.46 Right-Side View

3. Show also PIN part
4. **Hide** all the elements **not covered by PIN**

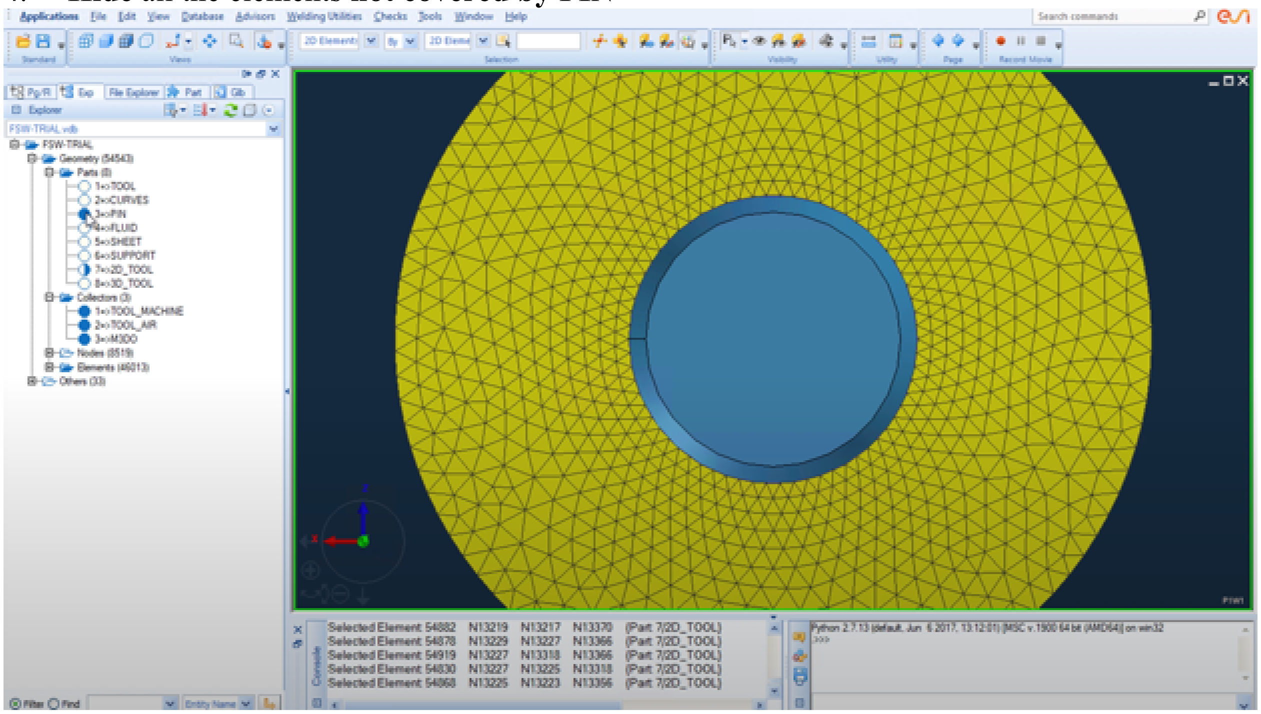


Figure 4.1.47 Hide the yellow elements

5. Select the **remaining elements** and **Add to New Collector**

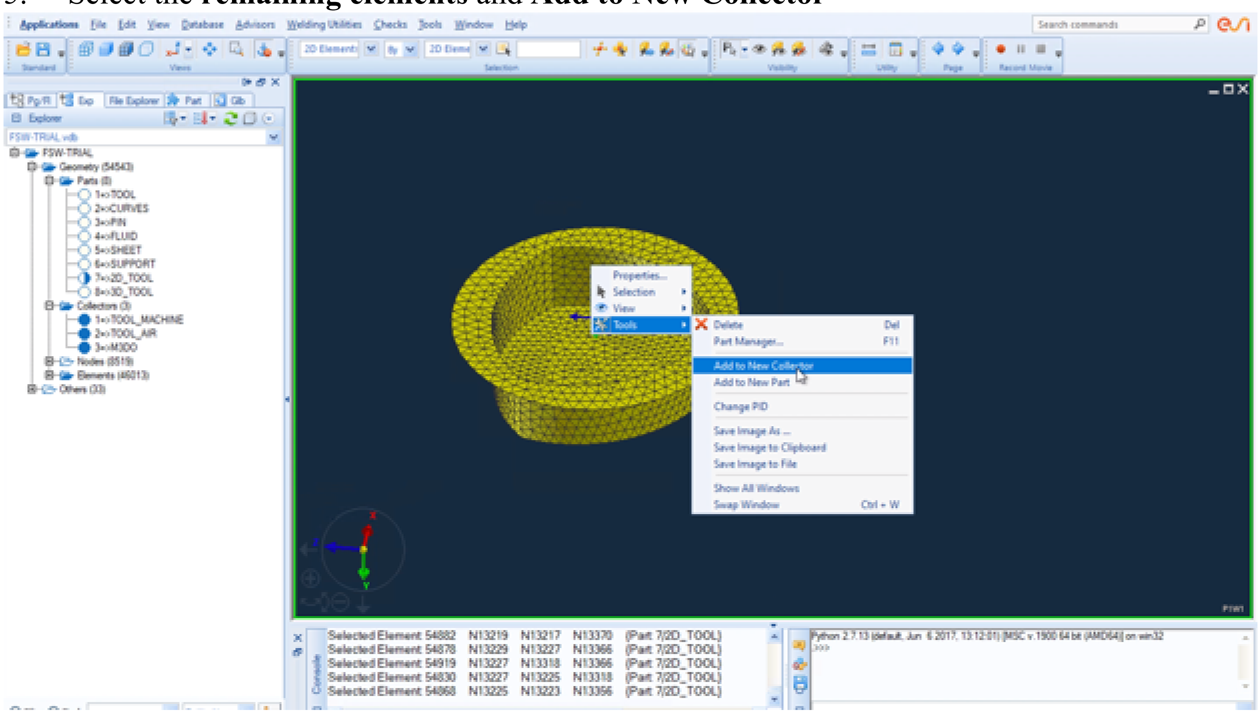


Figure 4.1.48 Collector_4

6. Select all the elements in **Collector_4**

7. Choose **Tools**, click **Transform**

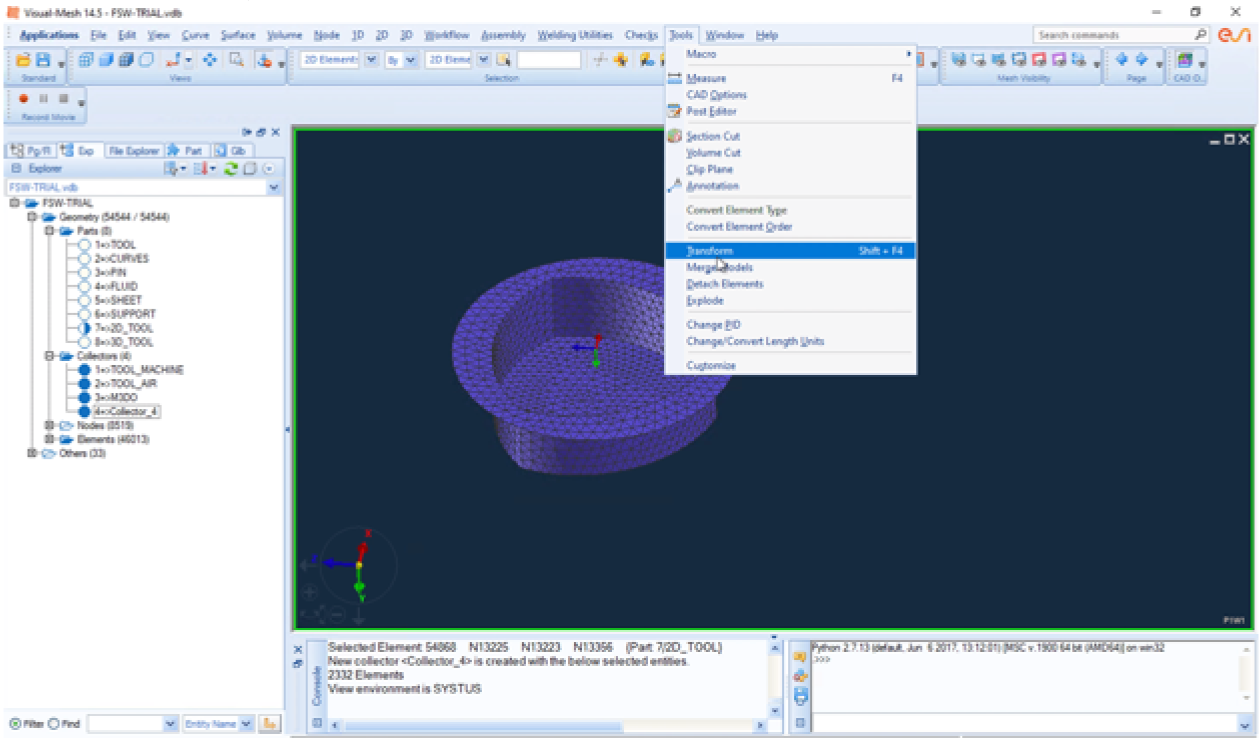


Figure 4.1.49 Transform

8. Choose **Copy**, input **Part 9**, click **Copy**

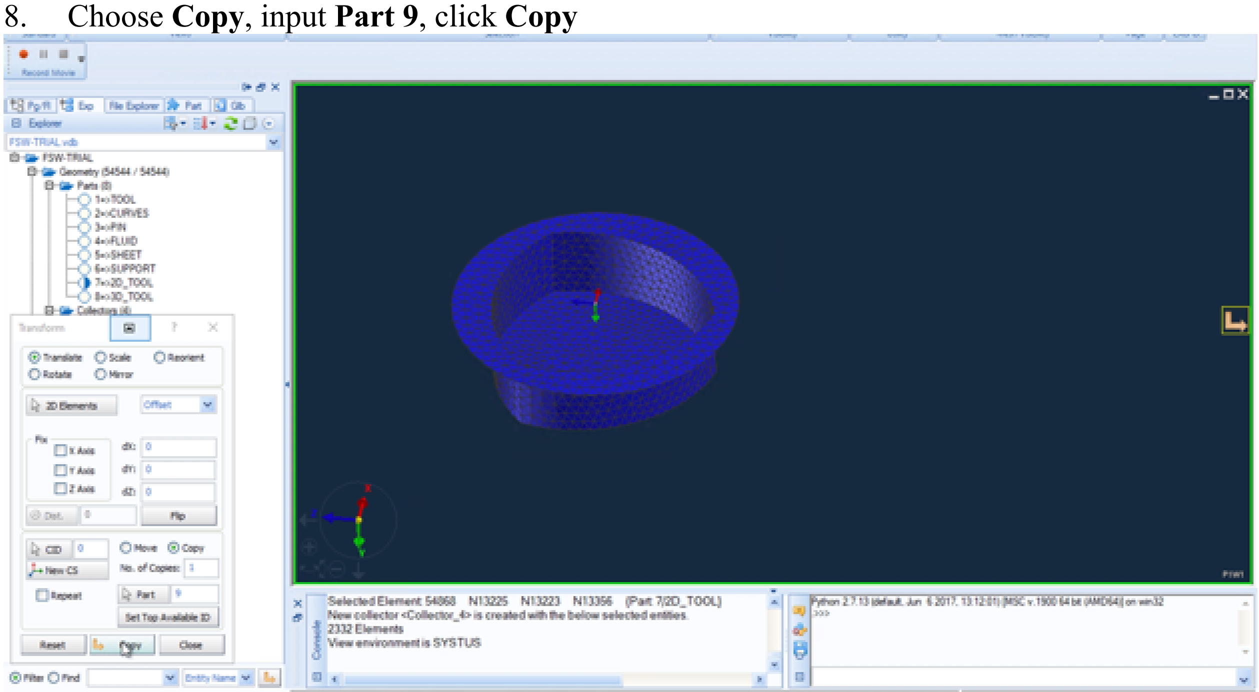


Figure 4.1.50 Transform

9. Rename **Collector_4** as **CT_M3DO_M3DP**
10. Choose **Checks**, click **Element Normals (Orientation)**

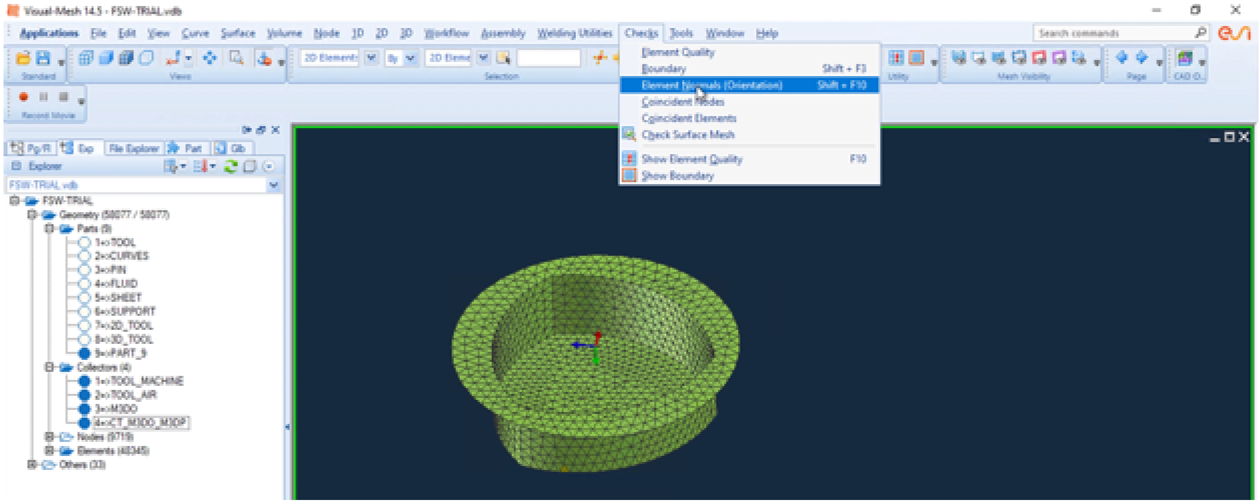


Figure 4.1.51 Check Orientation

11. Choose **Arrows**, click **Flip Normals**, choose **Elements**, select **all elements in PART_9**

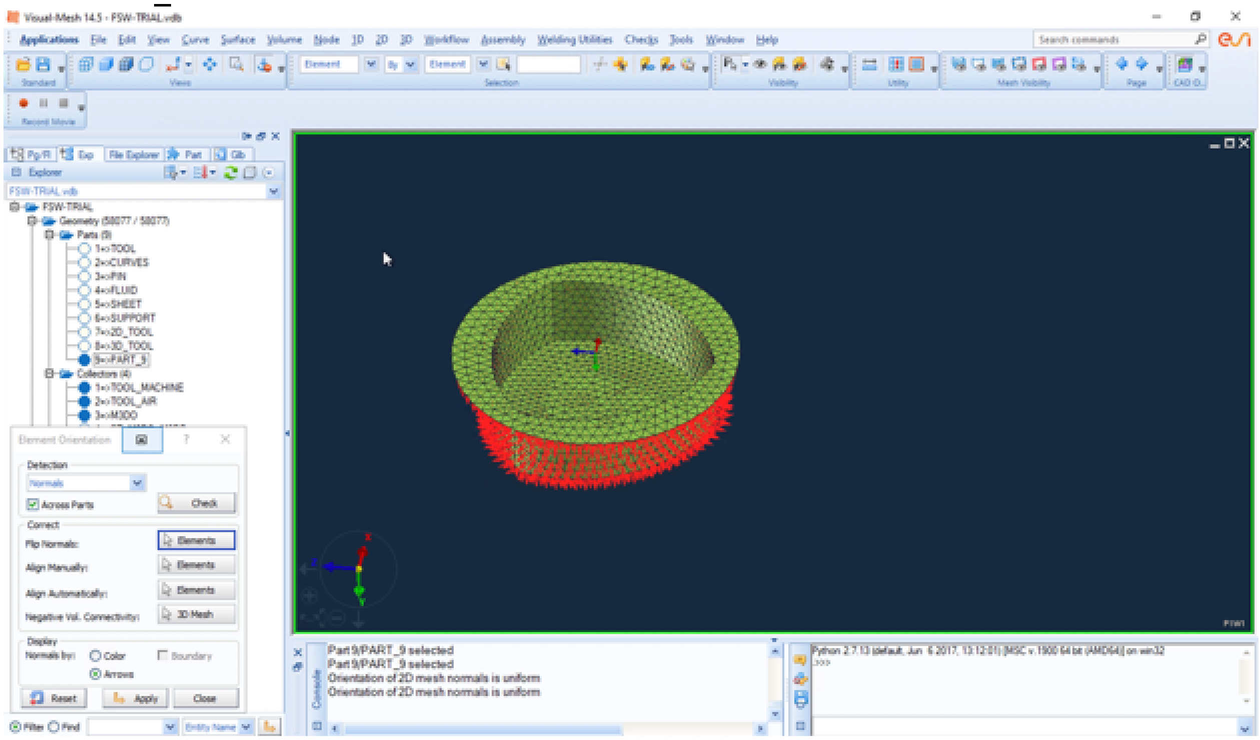


Figure 4.1.52 Flip Normals

12. Click **Apply**
13. Show **PIN** part
14. Choose **2D**, click **Automesh Surfaces**
15. Select **side surface** of **PIN** part

16. In **2D Mesh** window, **Set Element Size 0.3**,
17. In **Advanced** window, click **Within Part** and **Across Part**
18. In **Method** window, choose **Type Tria**
19. In **ID** window, input **Part 9**
20. Click **Create Mesh**, click **OK**

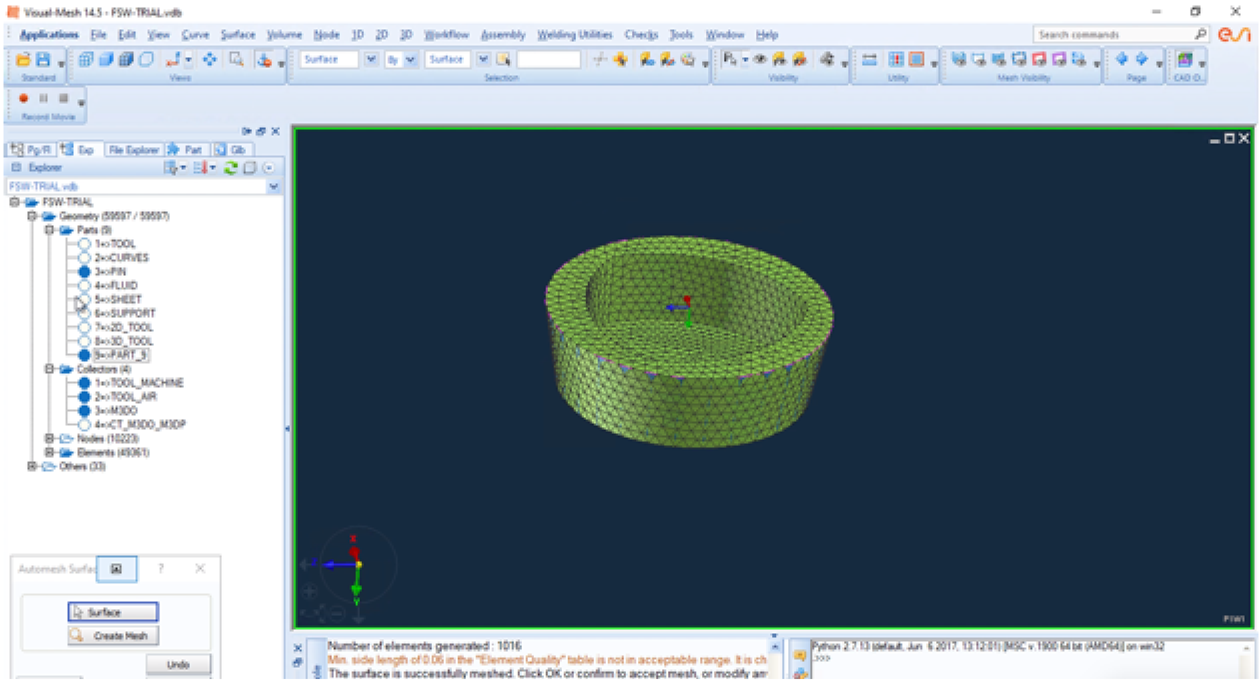


Figure 4.1.53 2D Mesh

21. Select t**he bottom face of PIN**
22. In **2D mesh** window, **Set Element Size 0.7**
23. Click **OK**

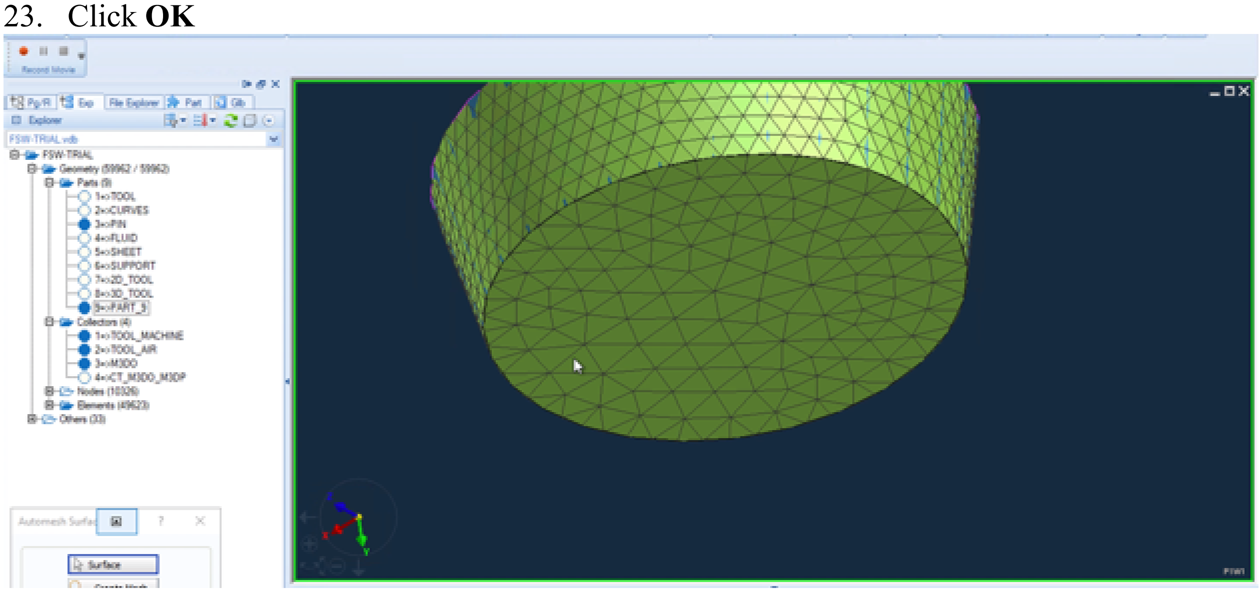


Figure 4.1.54 2D_PIN

24. Rename **PART_9** as **2D_PIN**
25. Choose **3D**, click **Tetra Mesh**
26. Select **all elements in 2D_PIN**, click **Apply**

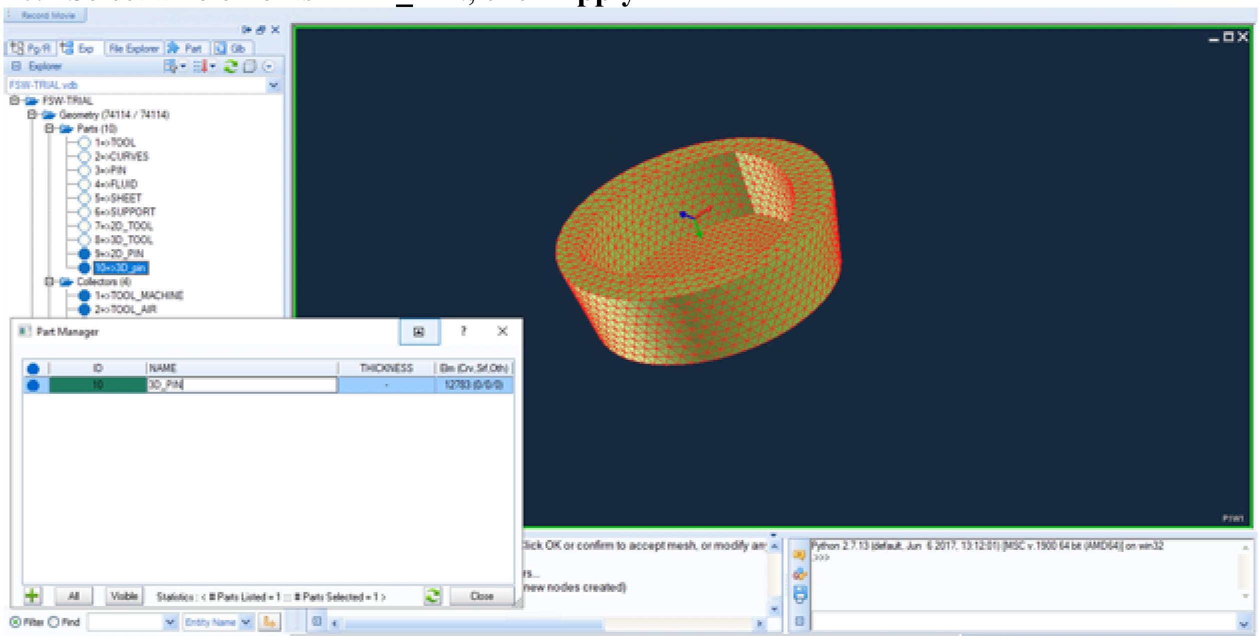


Figure 4.1.55 3D_PIN

27. Rename **PART_10** as **3D_PIN**

Define Collectors 5-7

1. Show **2D_PIN** part, hide **the outer surface**, show the same elements as **CT_M3DO_M3DP**
2. Select all the elements, **Add to New Collector**
3. Rename **Collector_5** as **CT_M3DP_M3DO**

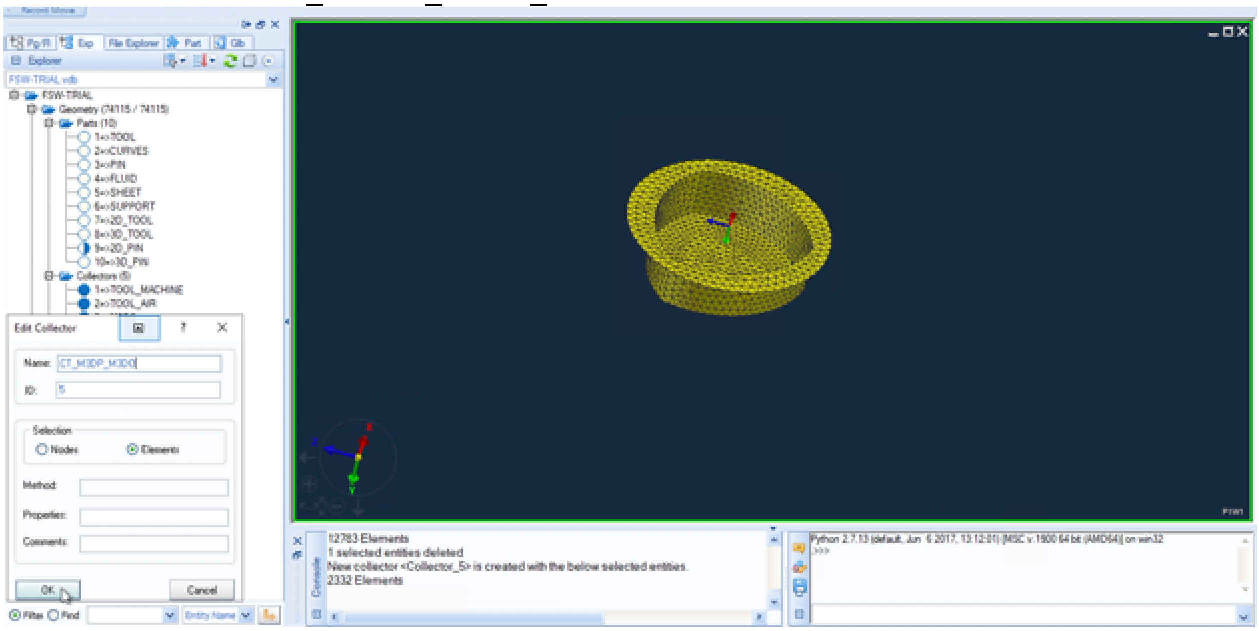


Figure 4.1.56 CT_M3DP_M3DO

4. Select all the elements in 3D_PIN, **Add to New Collector**

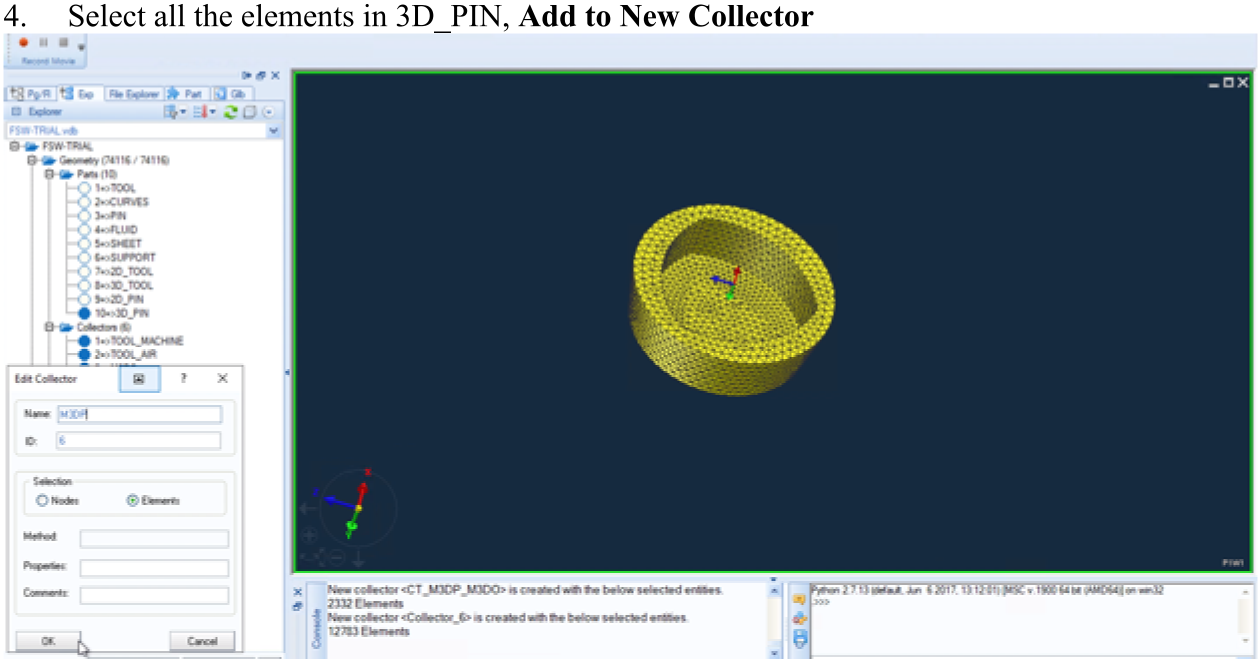


Figure 4.1.57 M3DP

5. Rename **Collector_6** as **M3DP**

6. In **Selection** window, choose **Node**

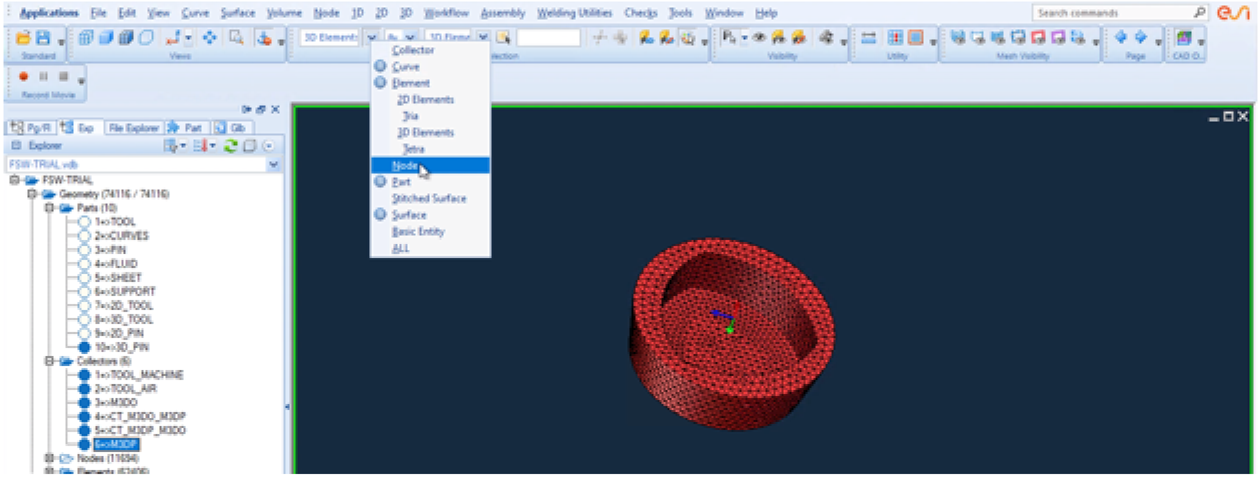


Figure 4.1.58 Selection Node

7. Select **all the nodes in 3D_PIN**, **Add to New Collector**
8. Rename **Collector_7** as **M3DP_N**

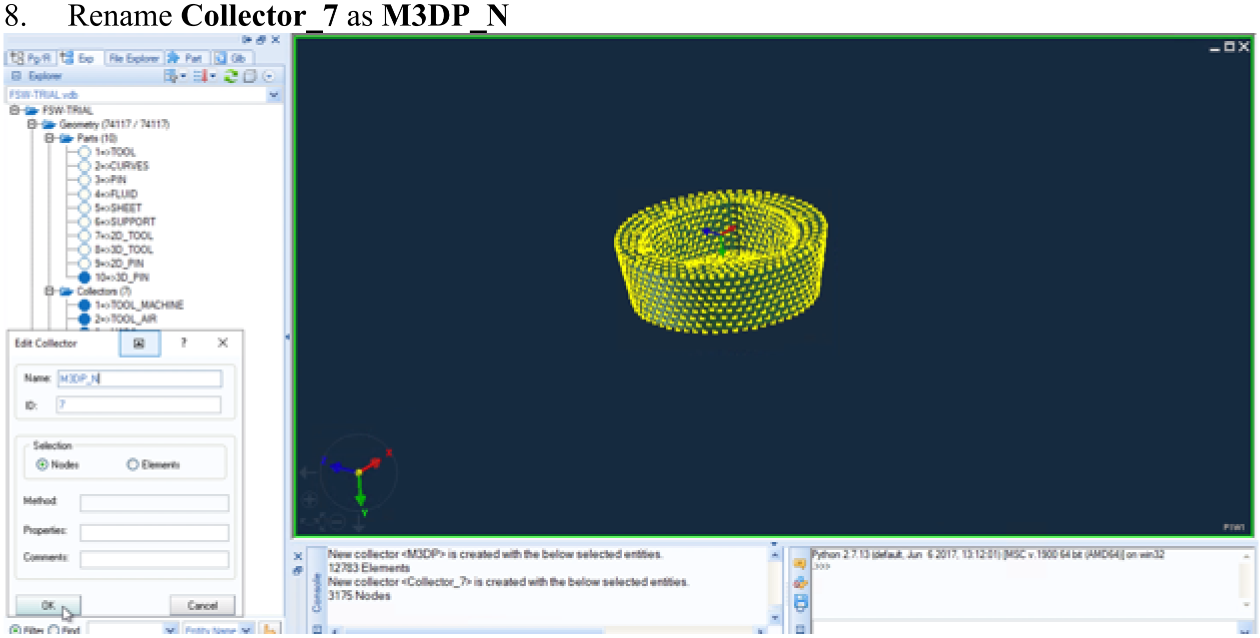


Figure 4.1.59 M3DP_N

Mesh FLUID Part & Collector 8-12

1. Show **FLUID** and **CURVES**
2. Choose **Surface,** click **Flat (Planar)**, select 2 circles, input **Part 4** and click **Apply**

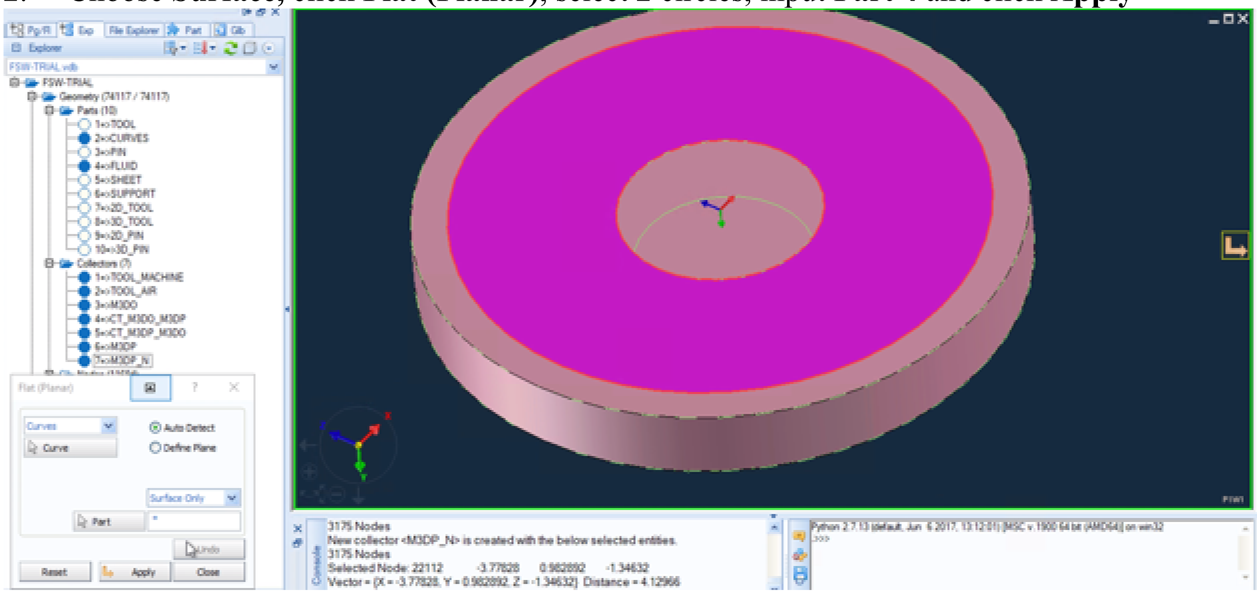


Figure 4.1.60 FLUID

3. Show only **FLUID** and **2D_PIN**
4. Choose **2D Element**, **Adjacent Element** and hide **Elements inside**.

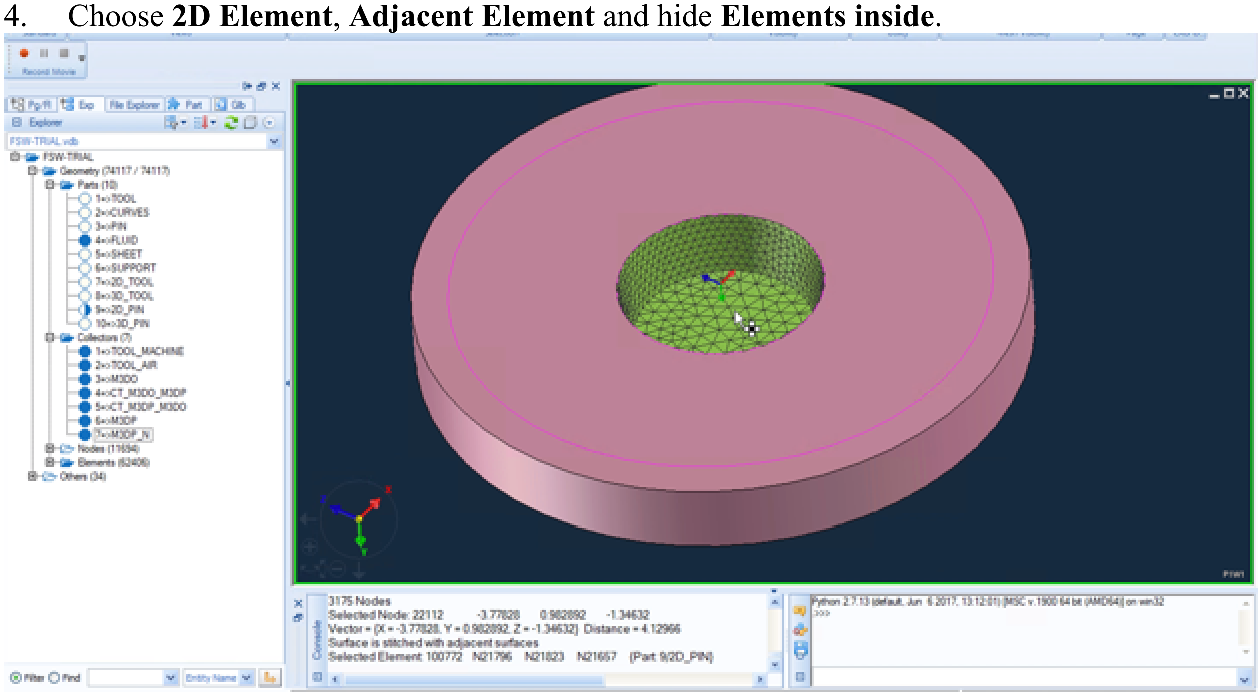


Figure 4.1.61 Part of 2D_PIN

5. Select the rest **Part of 2D_PIN**, **Add to New Collector**

6. Rename **Collector_8** as **CTM_M3DP_M3DF**

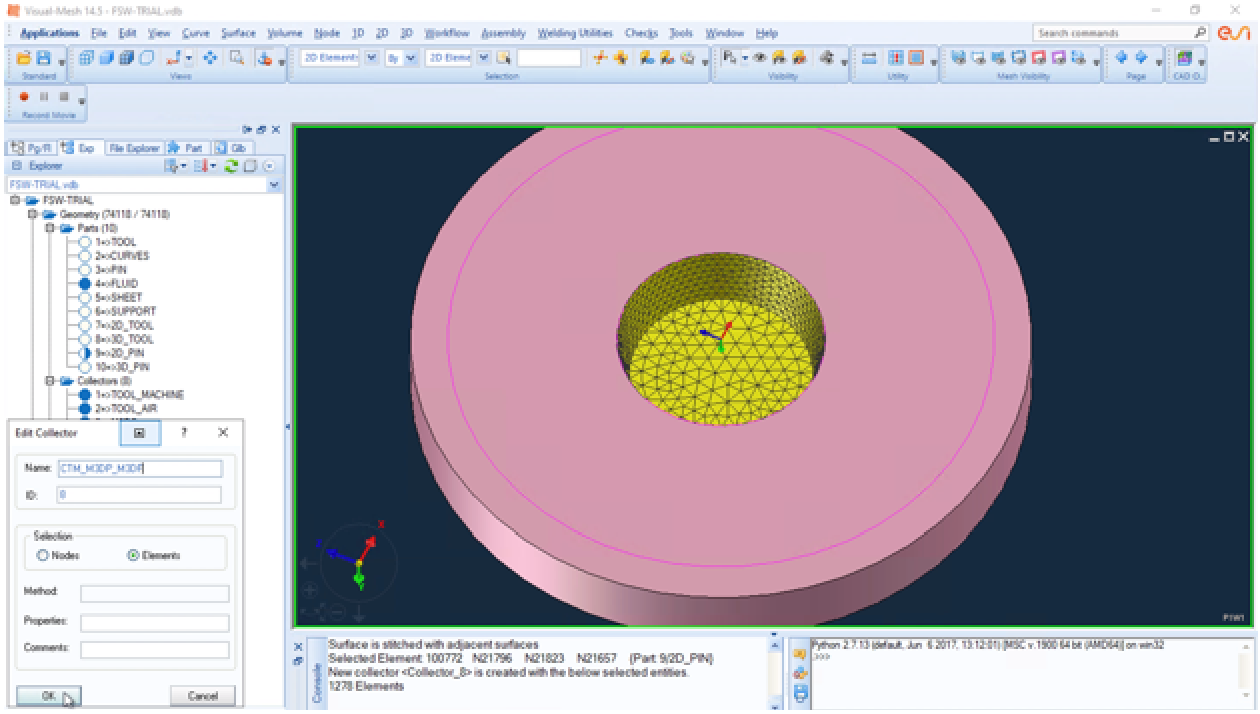


Figure 4.1.62 CTM_M3DP_M3DF

7. Choose **Tools**, click **Transform**
8. In **Transform** window, choose **Copy**, select all **2D Elements in CTM_M3DP_M3DF**
9. Input **Part 11**, click **Copy**

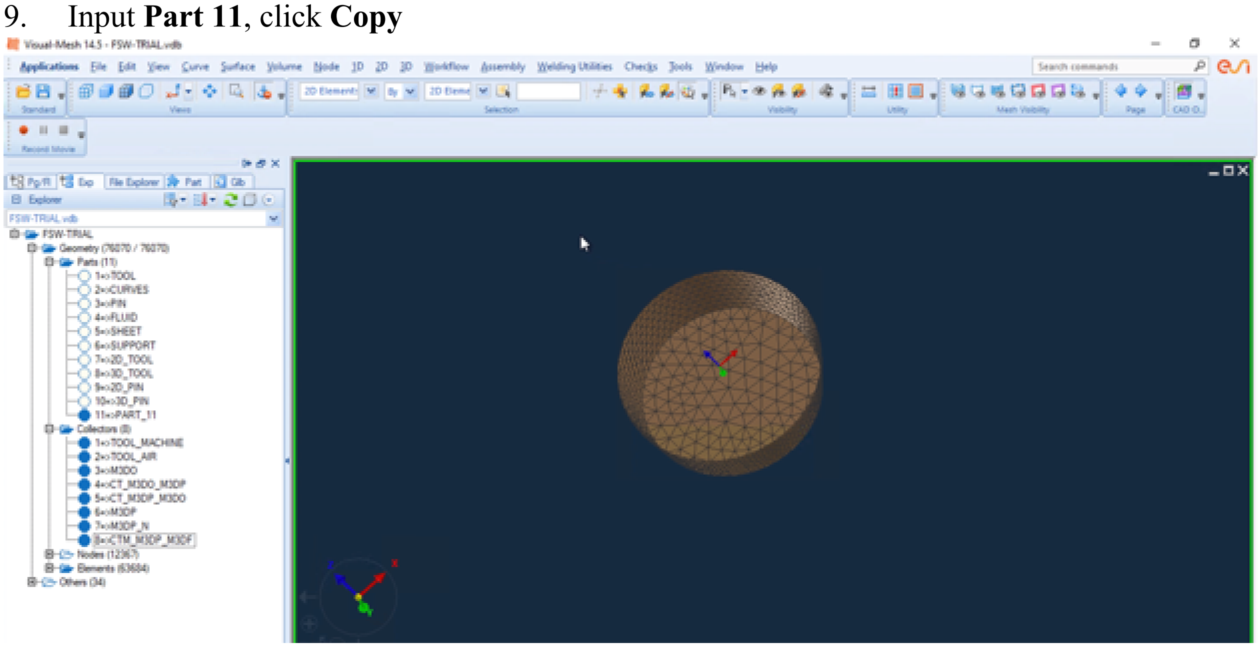


Figure 4.1.63 Part 11

10. Choose **Checks**, click **Element Normals (Orientation)**
11. Choose **Arrows**, click **Flip Normals**, choose **Elements**, select **all elements in PART_11**

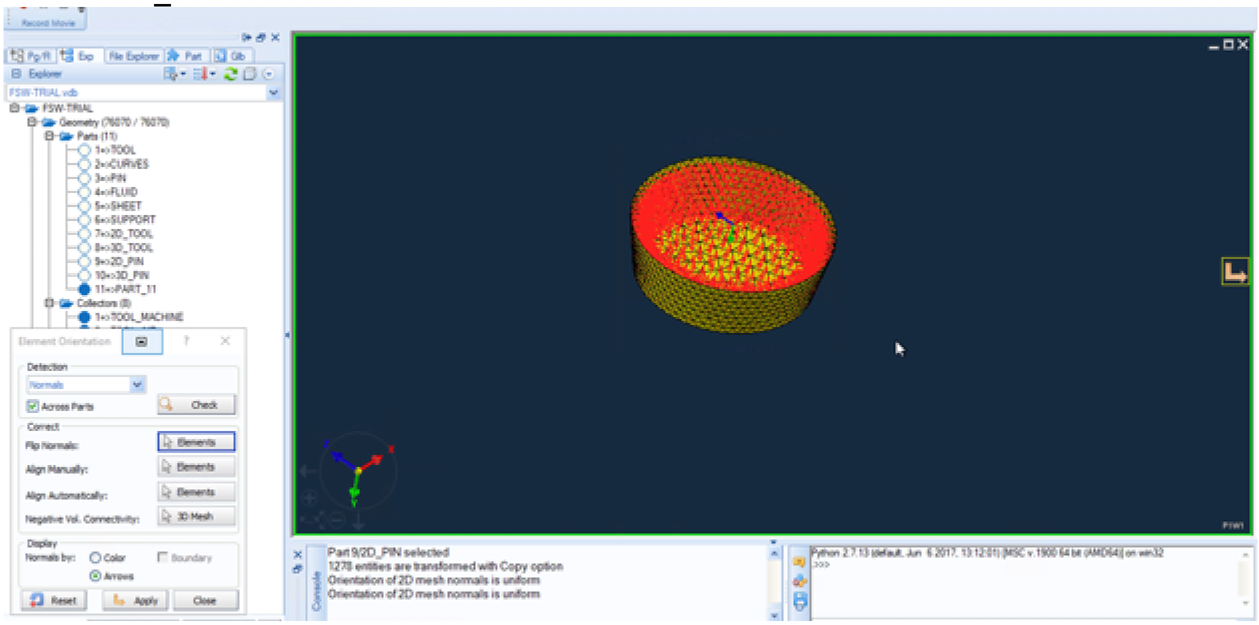


Figure 4.1.64 Flipped Normals of Part 11

12. Select **all elements in PART_11**, **Add to New Collector**
13. Rename **Collector_9** as **CTM_M3DF_M3DP**
14. Show **FLUID** and **PART_11**
15. Select **adjacent surface**, choose **2D**, click **Automesh Surfaces**
16. In **2D Mesh** window, **Set Element Size 0.6**, **Type Tria**, **Method Auto**, **Part 11**
17. Click **Create Mesh**, click **OK**

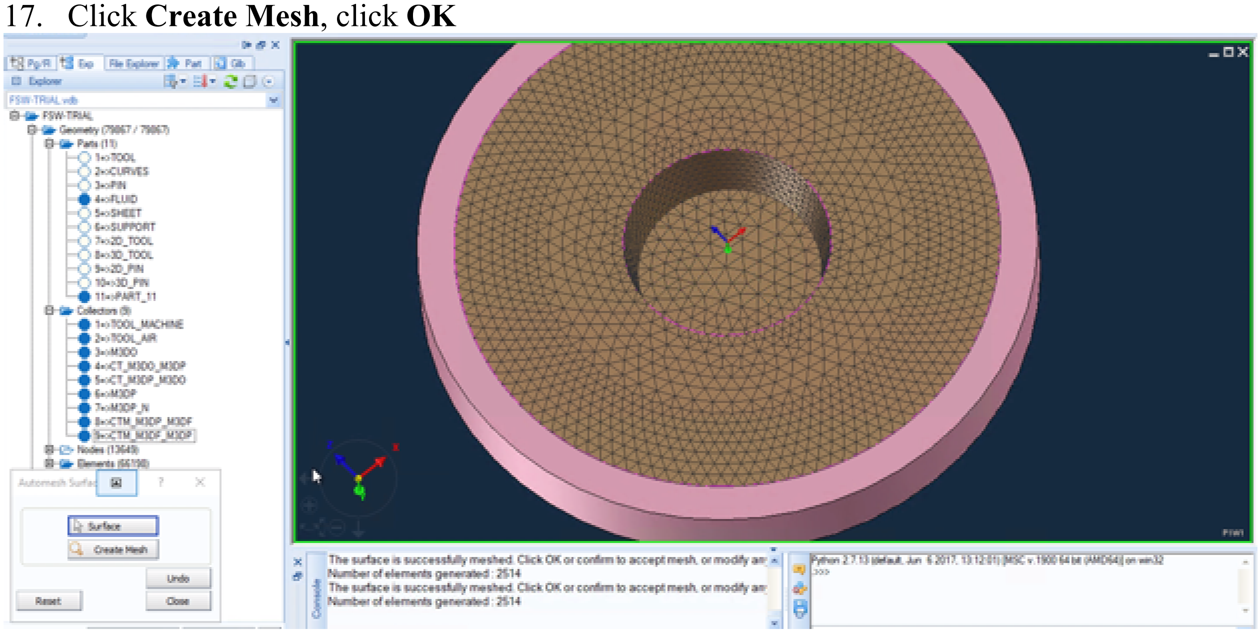


Figure 4.1.65 2D Mesh

18. Select **adjacent surface**, **Set Element Size 0.6**
19. Click **OK**

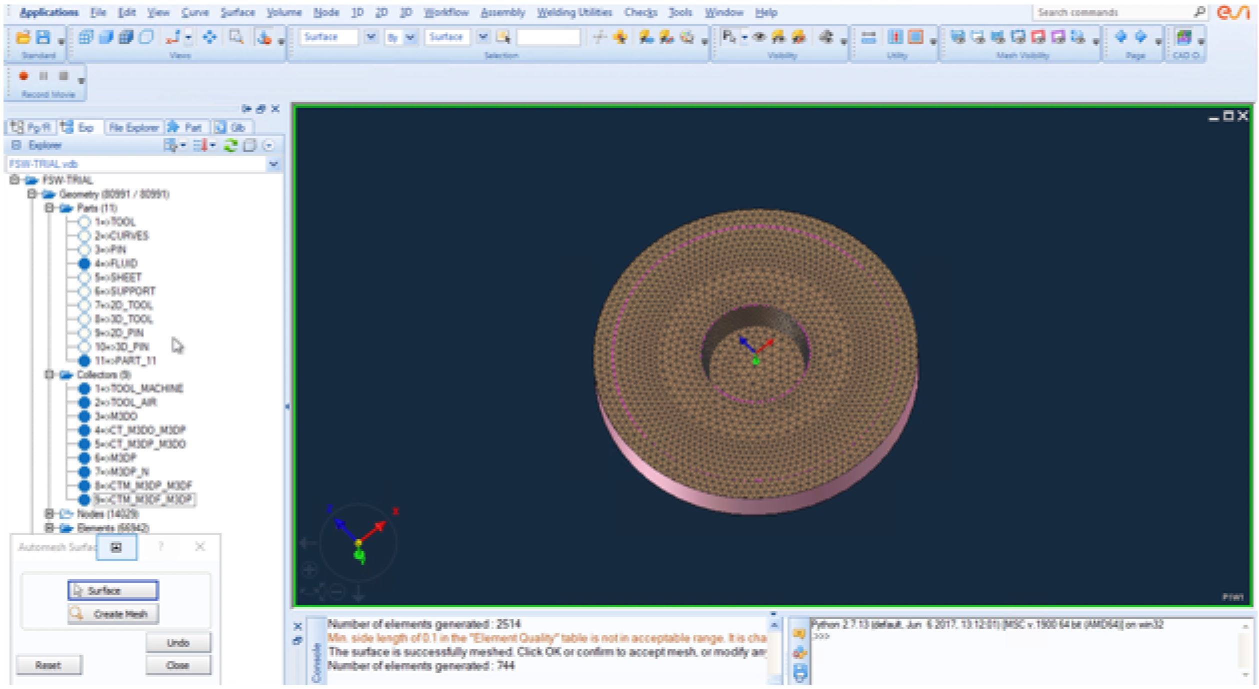


Figure 4.1.66 2D Mesh

20. Select **Side Surface**, **Set Element Size 0.7**
21. Click **OK**

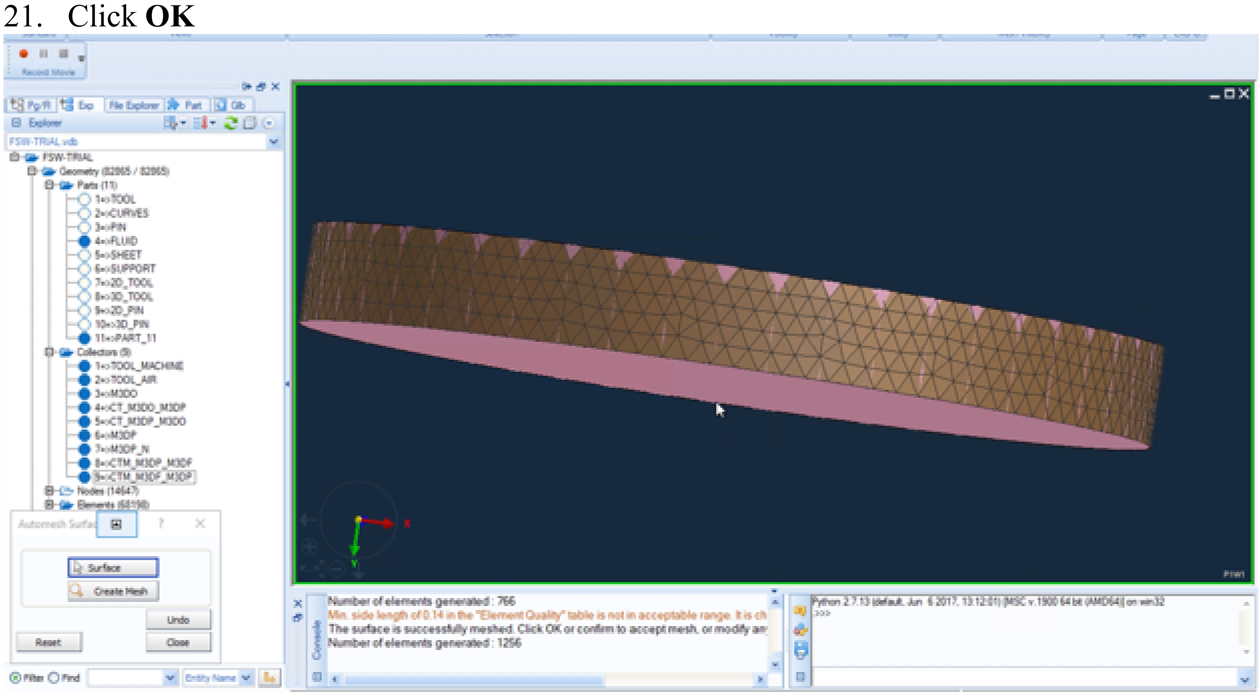


Figure 4.1.67 2D Mesh

22. Select **bottom surface**, **Set Element Size 1.2**
23. Click **OK**

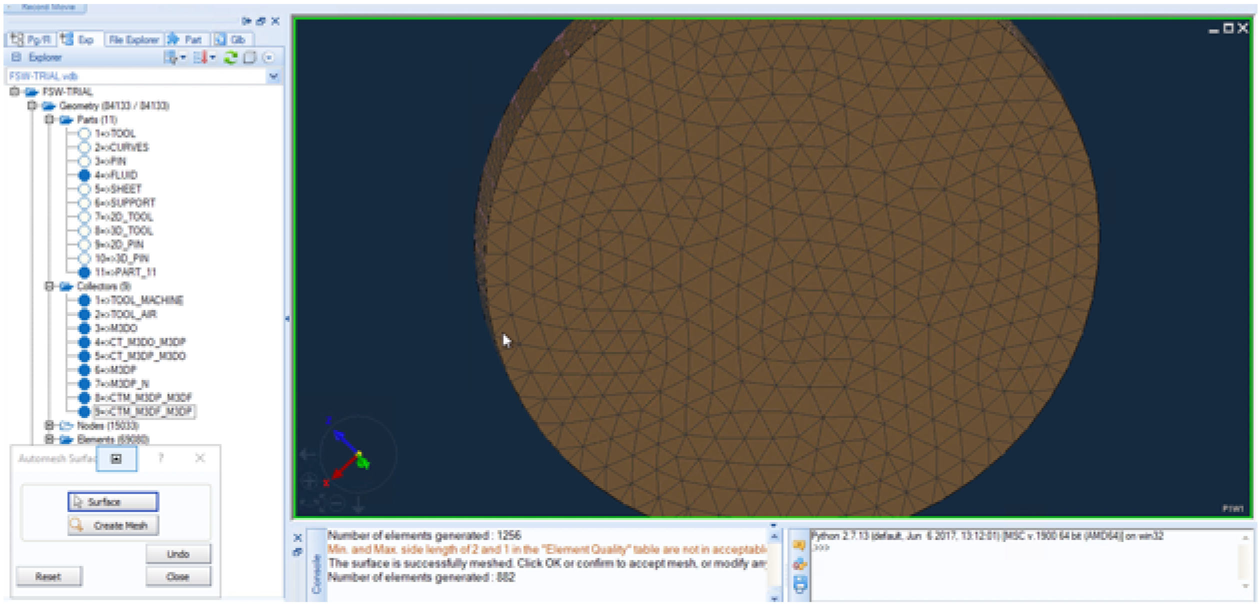


Figure 4.1.68 2D Mesh

24. Choose **3D**, click **Tetra Mesh**
25. In **Tetra Mesh** window, select **all elements in PART_11**
26. Click **Mesh**, click **Apply**

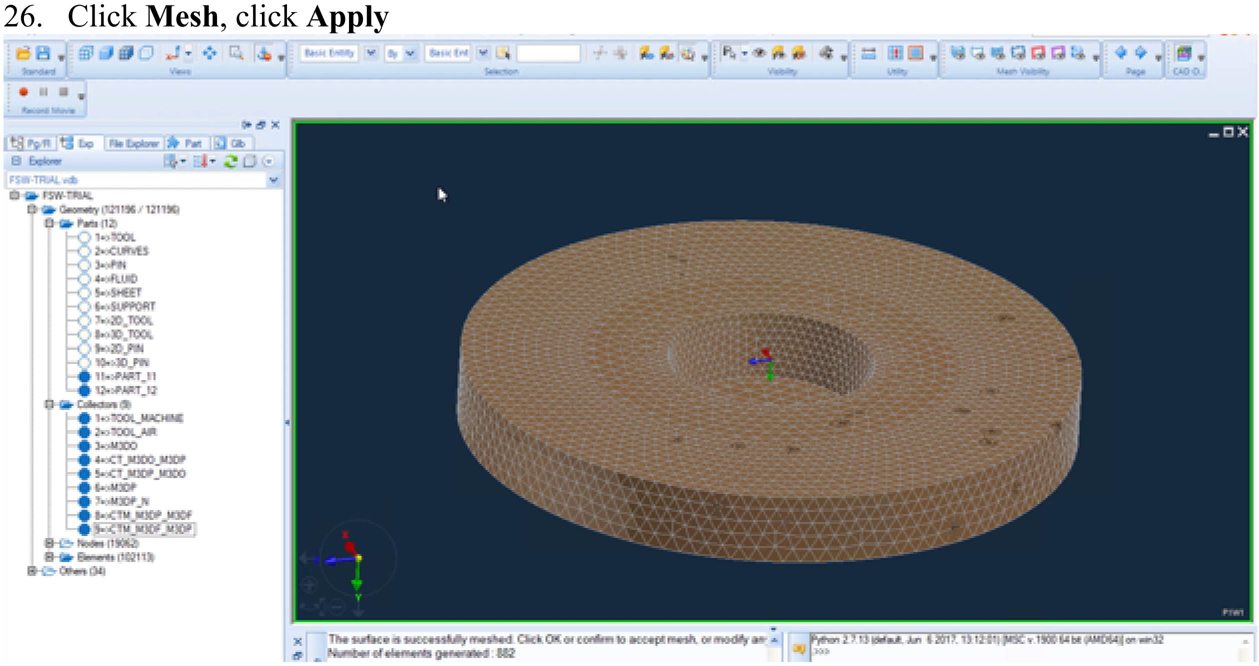


Figure 4.1.69 PART_12

27. Rename **PART_11** as **2D_FLUID**
28. Rename **PART_12** as **3D_FLUID**

29. Show **3D_FLUID**
30. ,Select **all 3D Elements**, **Add to New Collector**
31. Rename **Collector_10** as **M3DF**

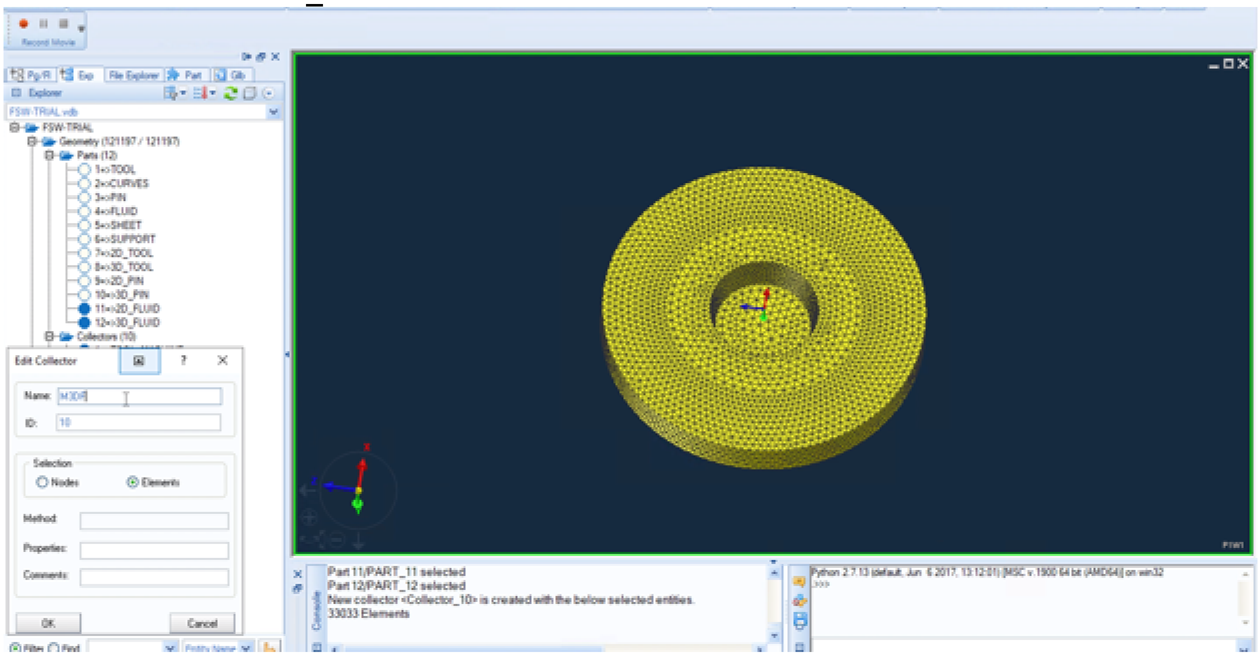


Figure 4.1.70 M3DF

32. Show **TOOL** and **2D_FLUID**
33. In **Views** window, choose **Left-Side View**. In **Selection** window, choose **Node**
34. Select **all the visible nodes in 2D_FLUID, except the outer-edge nodes**

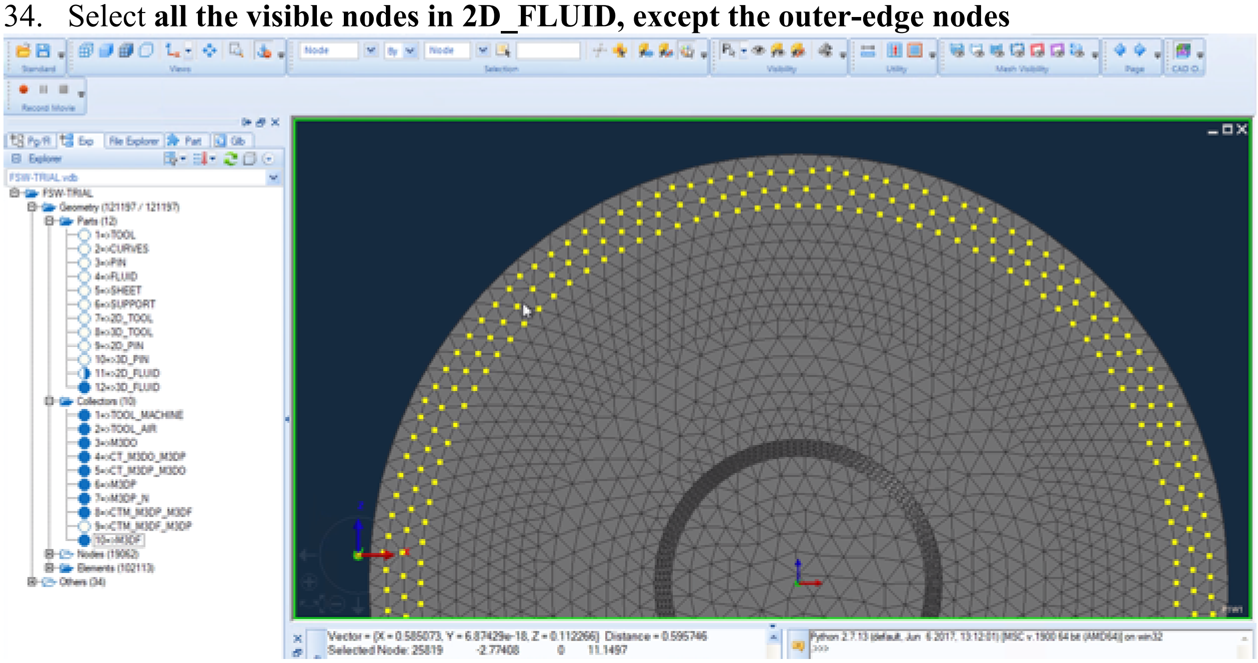


Figure 4.1.71 Nodes

35. **Add to New Collector** your node selection
36. Partly show the bottom surface of **2D_FLUID** and **Collector_11**
37. Select **all nodes in bottom surface**

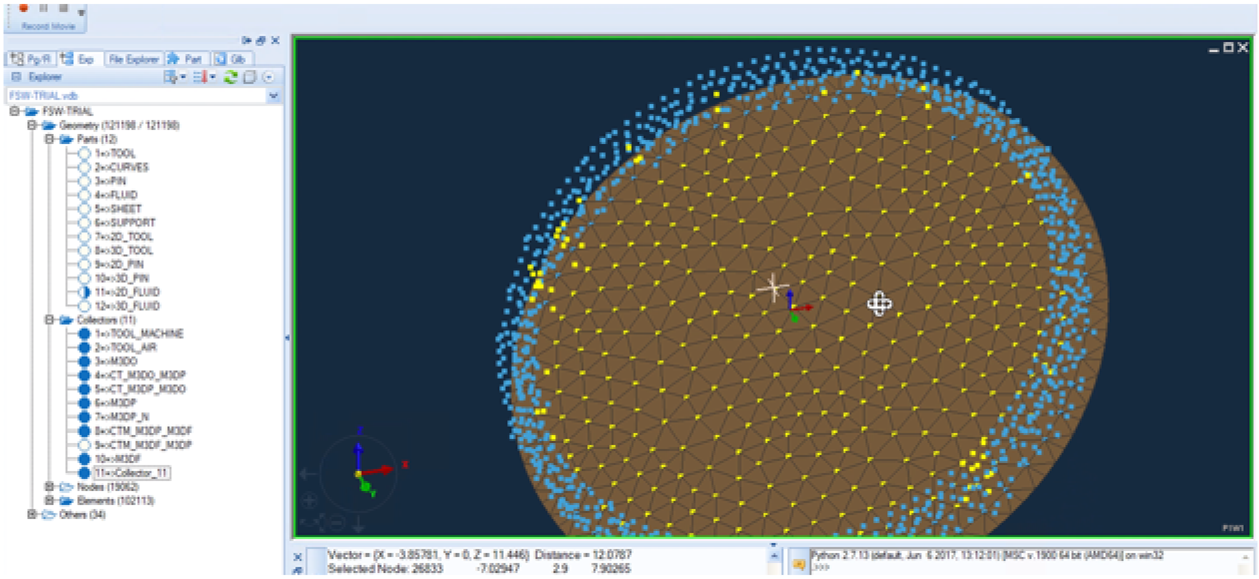


Figure 4.1.72 Nodes

38. Right click on **11=>Collector_11**
39. Click **Add Selection**

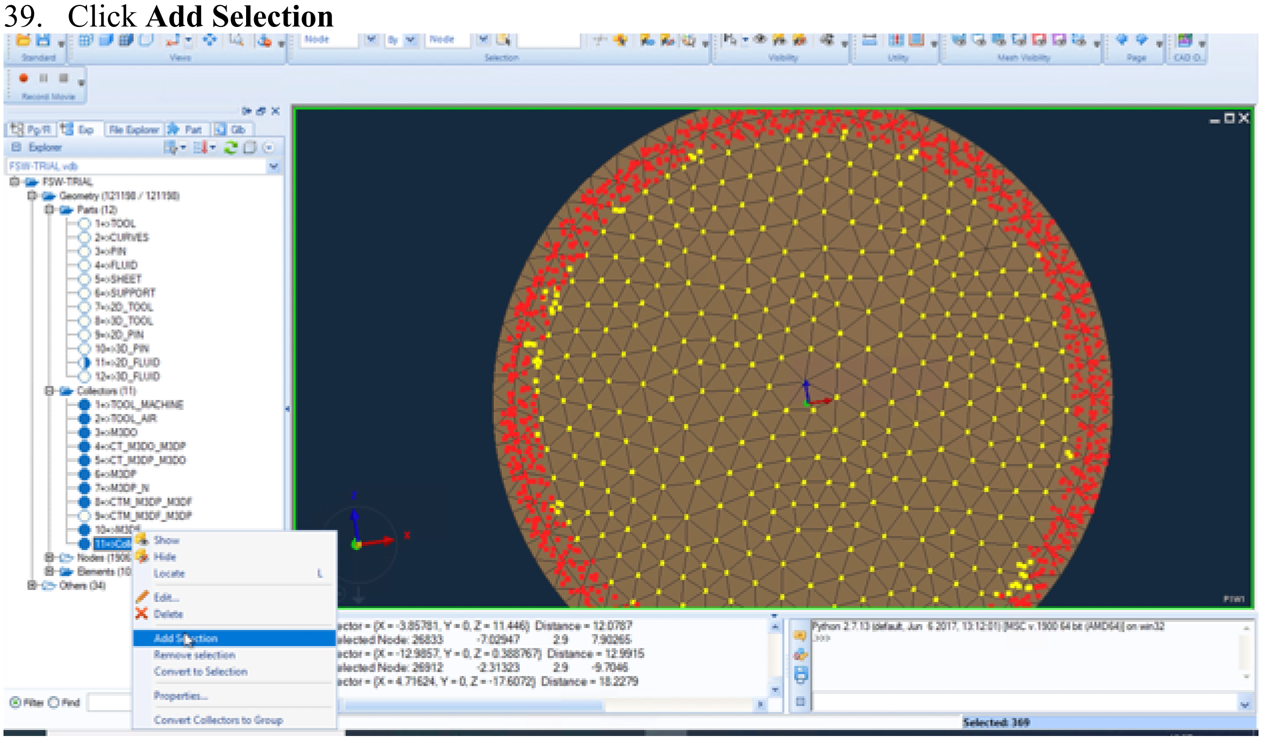


Figure 4.1.73 Collector_11

40. Rename **Collector_11** as **VZ**

41. Show only **the side surface of 2D_FLUID**
42. In **Views** window, click **Front View**. **Selection Node**
43. Select **all nodes** in the figure, **Add to New Collector**

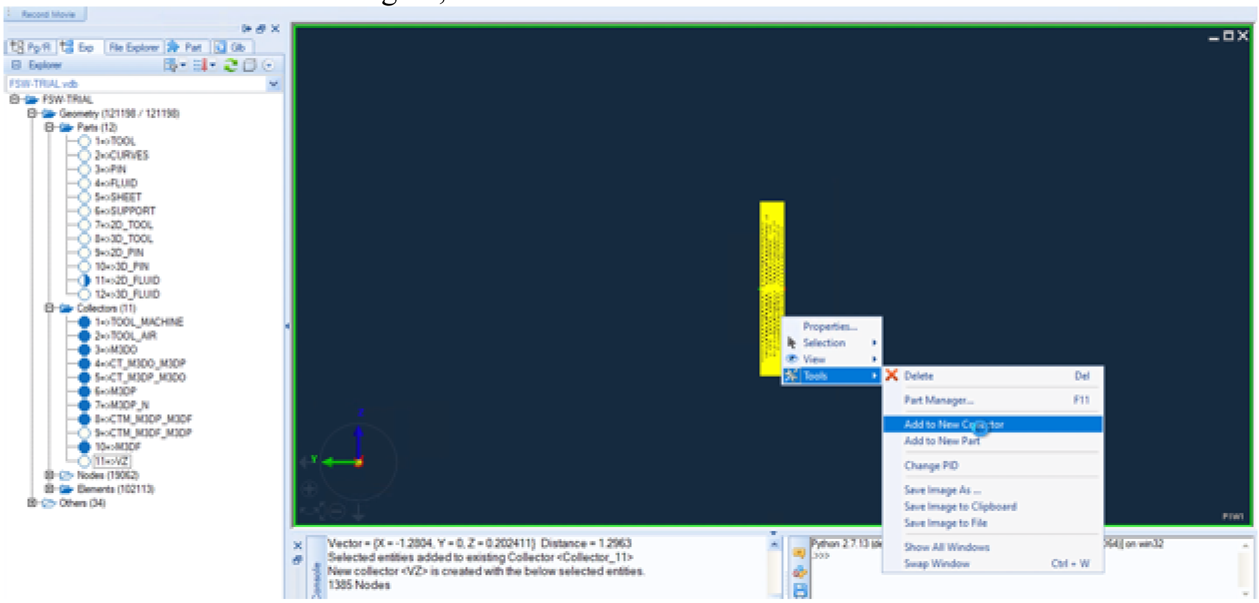


Figure 4.1.74 Collector_12

44. Rename **Collector_12** as **V_WELDING**

Mesh SHEET & Collector 13-14

1. Show **CURVES**, **SHEET** and **2D_FLUID**
2. Choose **Surface**, click **Split**
3. Select **sheet top surface**, **Split by Curves**, Select **the curve of fluid edge**
4. Click **Apply**

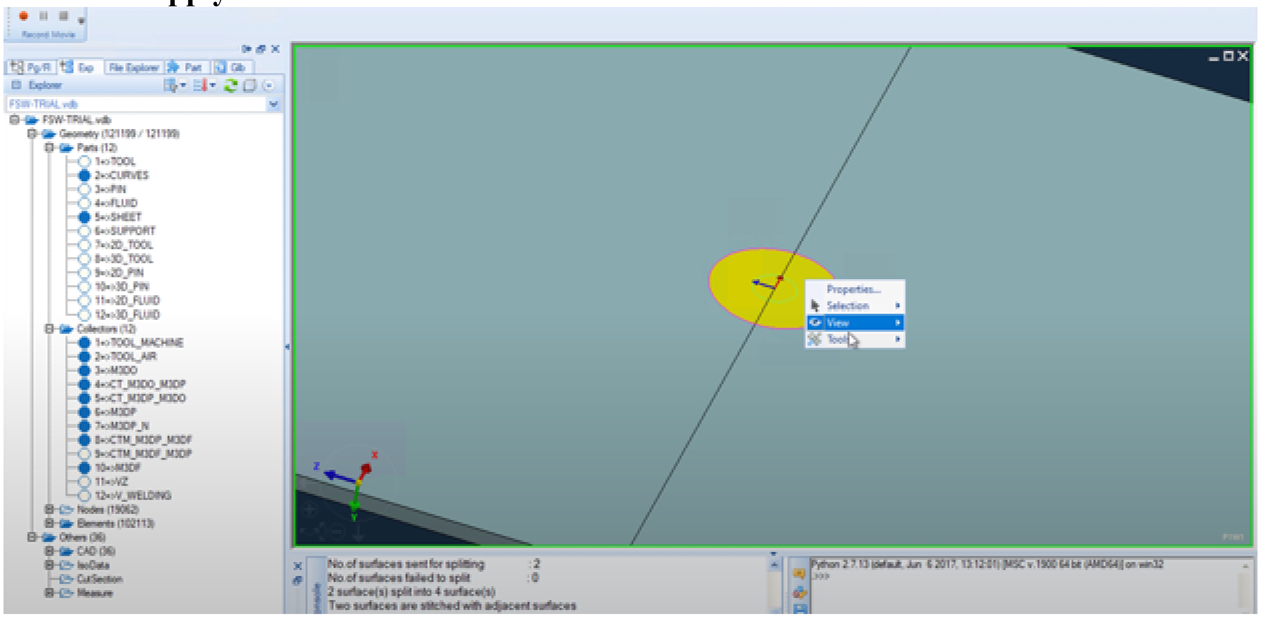


Figure 4.1.75 Split top surface

5. Delete the selected **yellow area**
6. Choose **Surface**, click **Split**
7. In **Surface Split** window, Select the whole **SHEET part**, **Split by Plane**
8. In **Plane Definition** window, choose **Standard Axis**, **Along Z Axis**, **Base Pt 0 0 70**
9. Click **Apply**

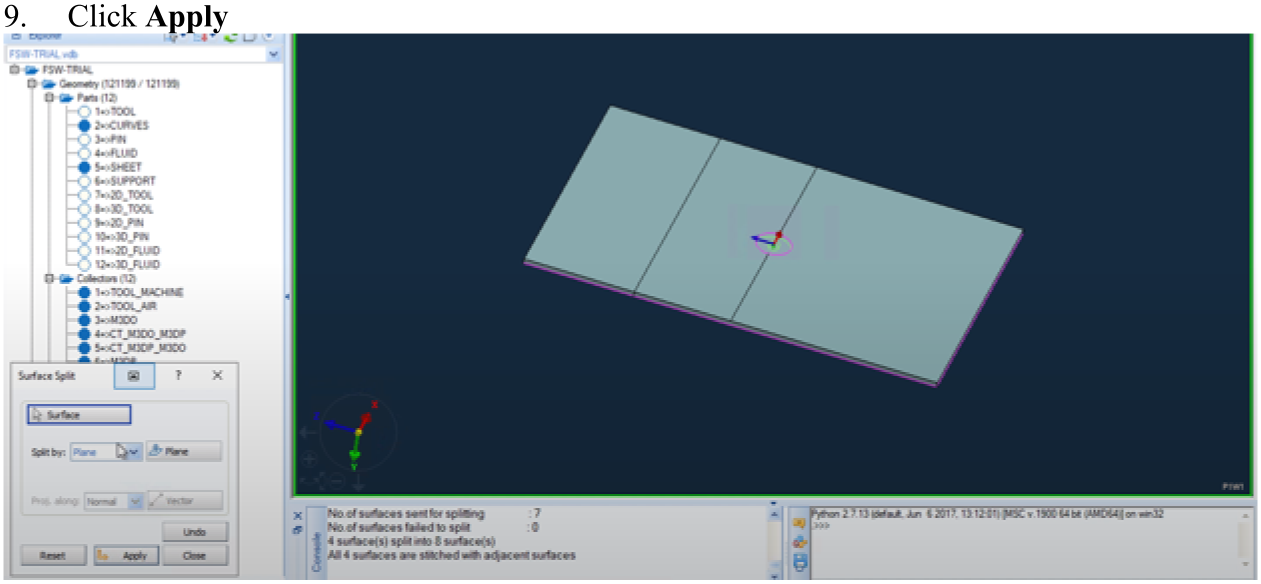


Figure 4.1.76 Split top surface

10. Select the whole **SHEET part**
11. **Base Pt 0 0 25**, click **OK**, click **Apply**
12. **Base Pt 0 0 -25**, click **OK**, click **Apply**
13. **Base Pt 0 0 -70**, click **OK**, click **Apply**

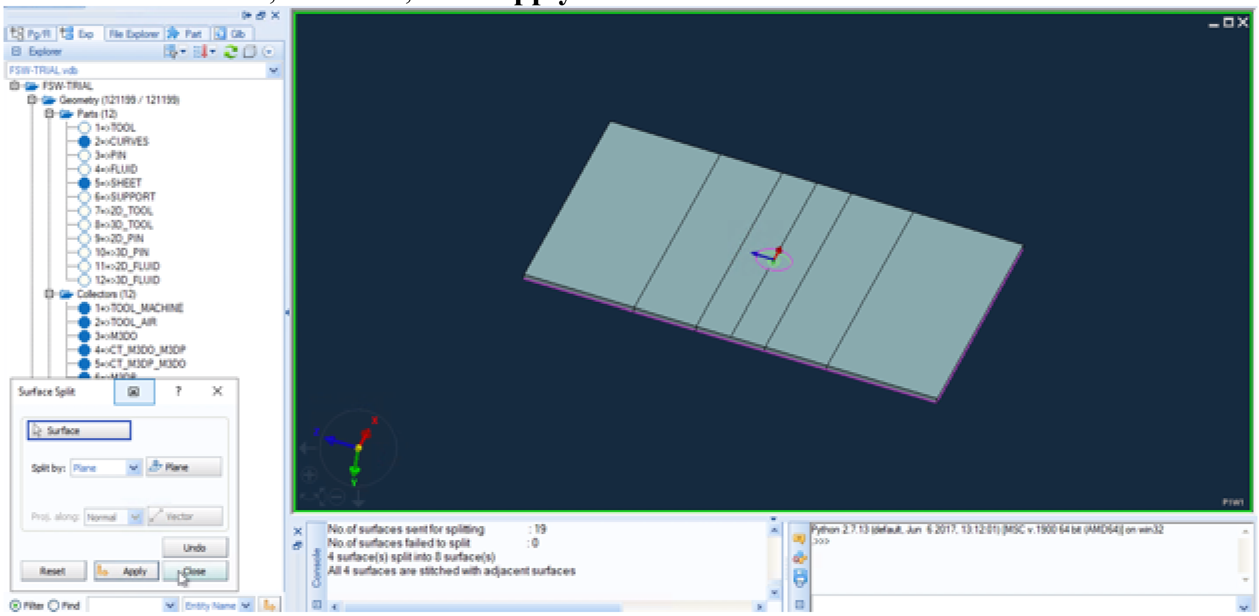


Figure 4.1.77 Split top surface

14. Show **outer surface of M3DF**, show **SHEET**
15. Choose **2D**, click **Automesh Surface**, Select the adjacent surface
16. **Set Element Size 1.5**, **Type Tria**, **Method Auto**, **Part 13**
17. Click **OK**

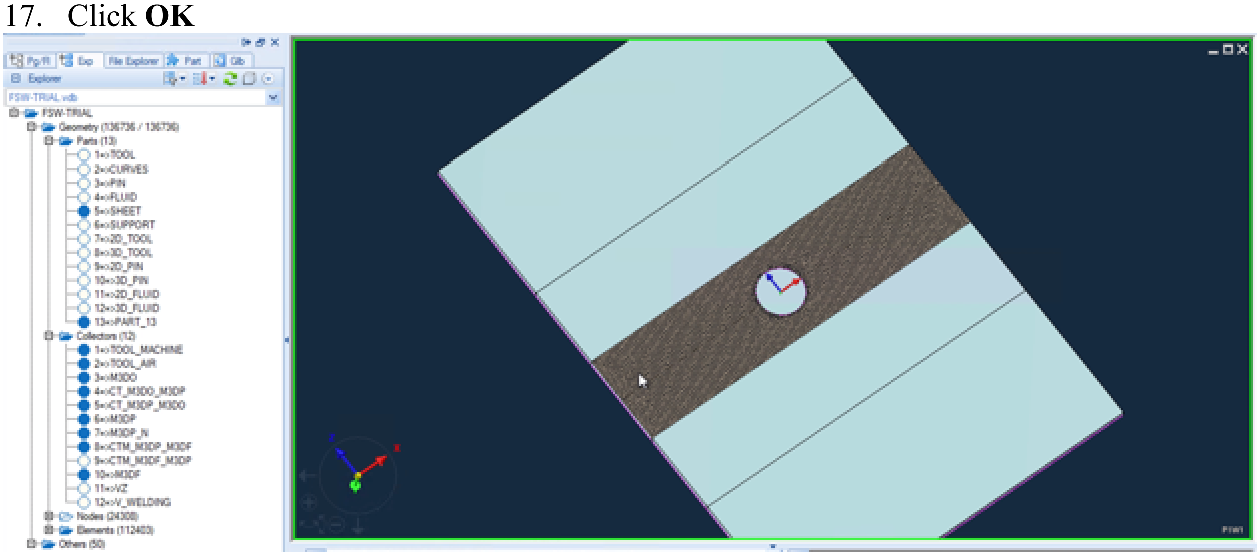


Figure 4.1.78 2D Mesh

18. Select the two adjacent surfaces

19. **Set Element Size 2.5**, reduce edge nodes number from **80 to 60**
20. Click **OK**

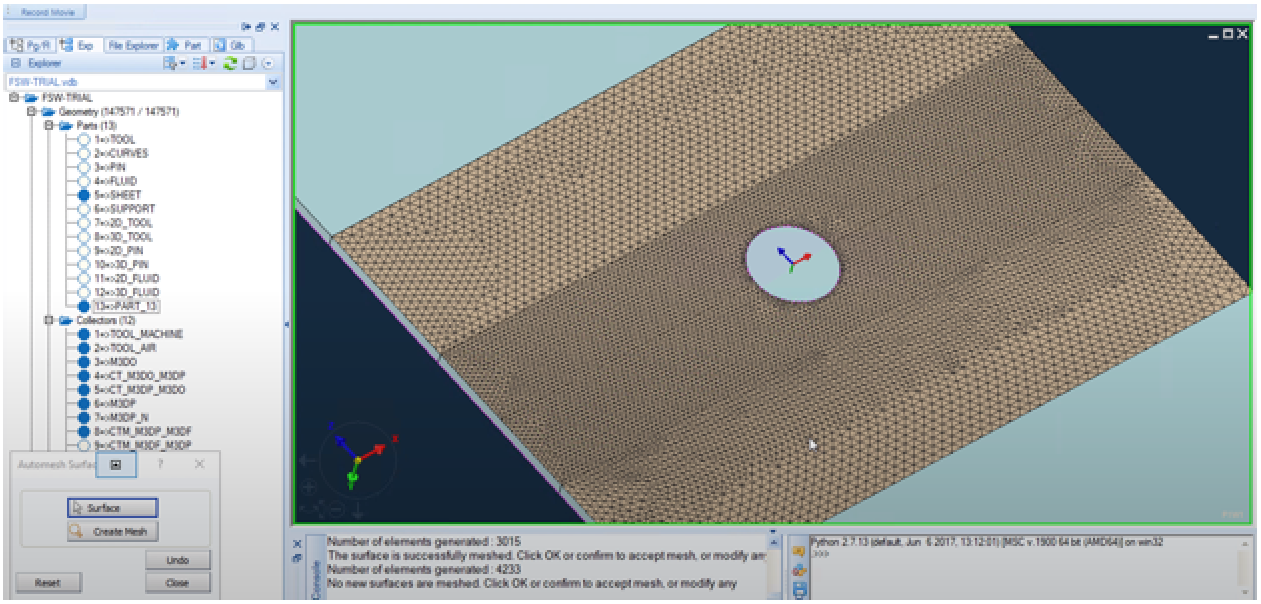


Figure 4.1.79 2D Mesh

21. Select two adjacent surfaces, **Set Element Size 5**, reduce edge nodes number from **40 to 30**
22. Click **OK**

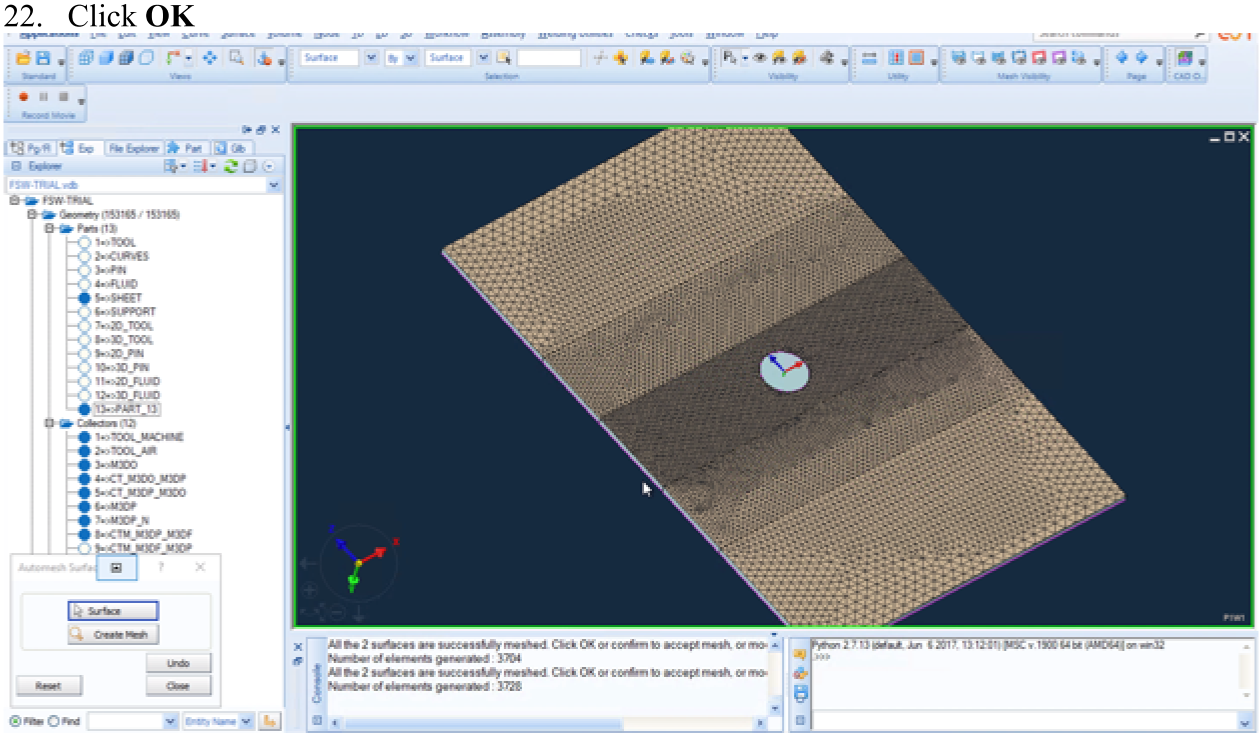


Figure 4.1.80 2D Mesh

23. Select 2 middle side surfaces
24. **Set Element Size 1**, reduce edge number **51 to 34**, **Method Map**
25. Click **OK**

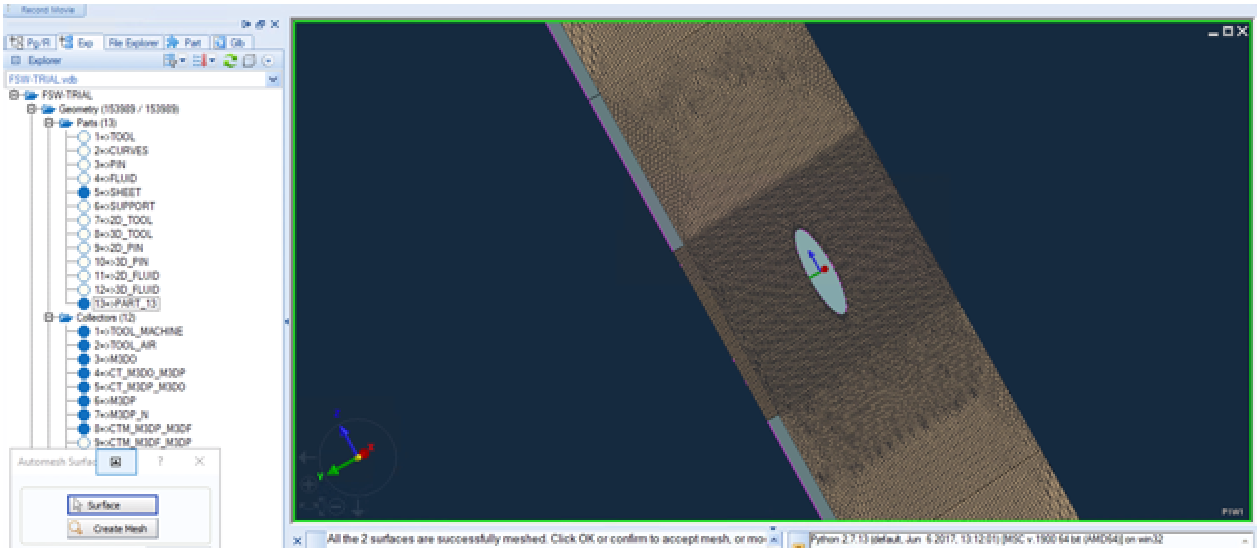


Figure 4.1.81 2D Mesh

26. Select 4 adjacent side surface
27. **Set Element Size 1**, reduce edge number **46 to 18**, **Method Map**
28. Click **OK**
29. Select 4 adjacent side surface
30. **Set Element Size 1**, reduce edge number **81 to 16**, **Method Map**
31. Click **OK**

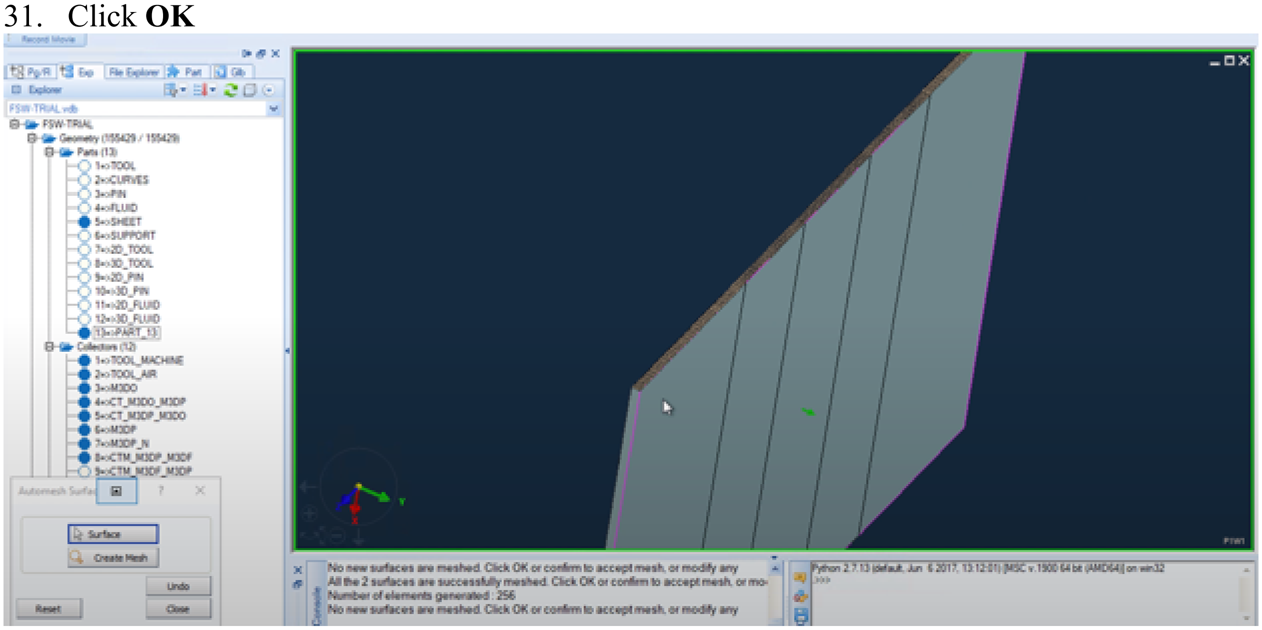


Figure 4.1.82 2D Mesh

32. Select bottom surface
33. **Set Element Size 1.5**, **Method Auto**
34. Click **OK**
35. Select the two adjacent surfaces
36. **Set Element Size 2.5**, reduce edge nodes number from **80 to 60**
37. Click **OK**
38. Select two adjacent surfaces, **Set Element Size 5**, reduce edge nodes number from **40 to 30**
39. Click **OK**
40. Select the remaining 2 surfaces, **Set Element Size 1**
41. Click **OK**

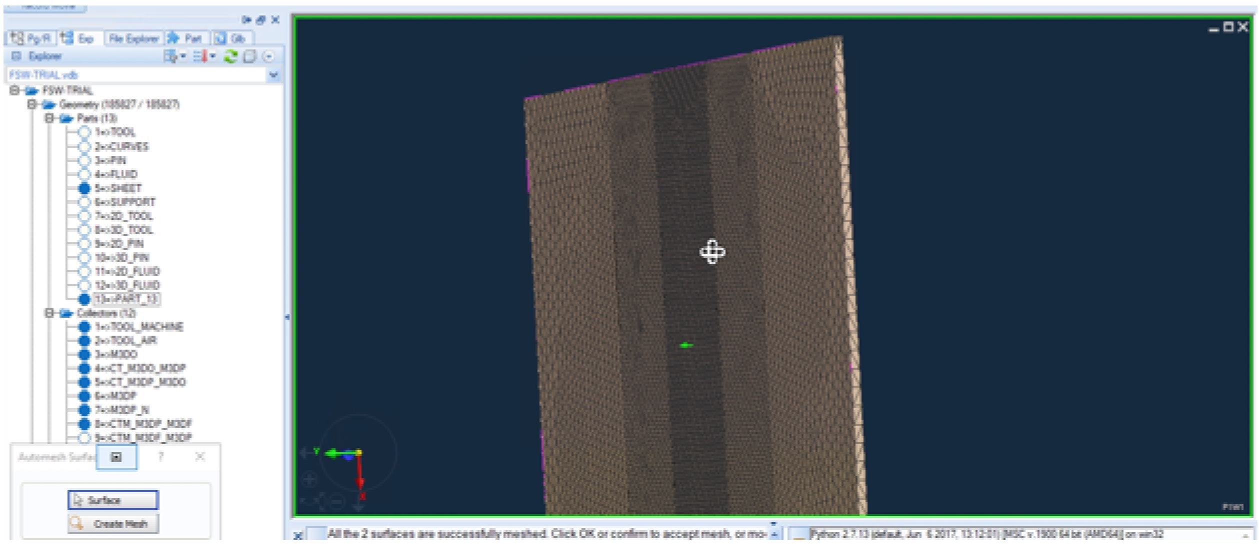


Figure 4.1.83 PART_13

42. Choose **3D**, click **Tetra Mesh**
43. Select all elements in **PART_13** and **outer surface of 2D_FLUID**
44. Click **Mesh**, click **Apply**

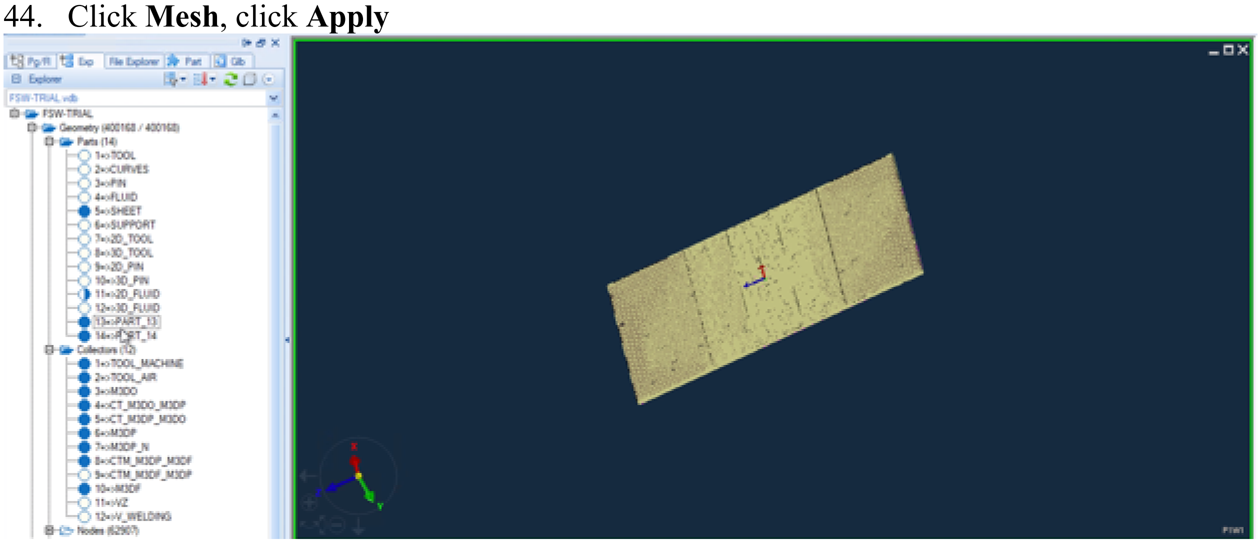


Figure 4.1.84 PART_14

45. Rename **PART_13** as **2D_SHEET**
46. Rename **PART_14** as **3D_SHEET**
47. Select all elements in **3D_SHEET**, **Add to New Collector**
48. Rename **Collector_13** as **M3DT**

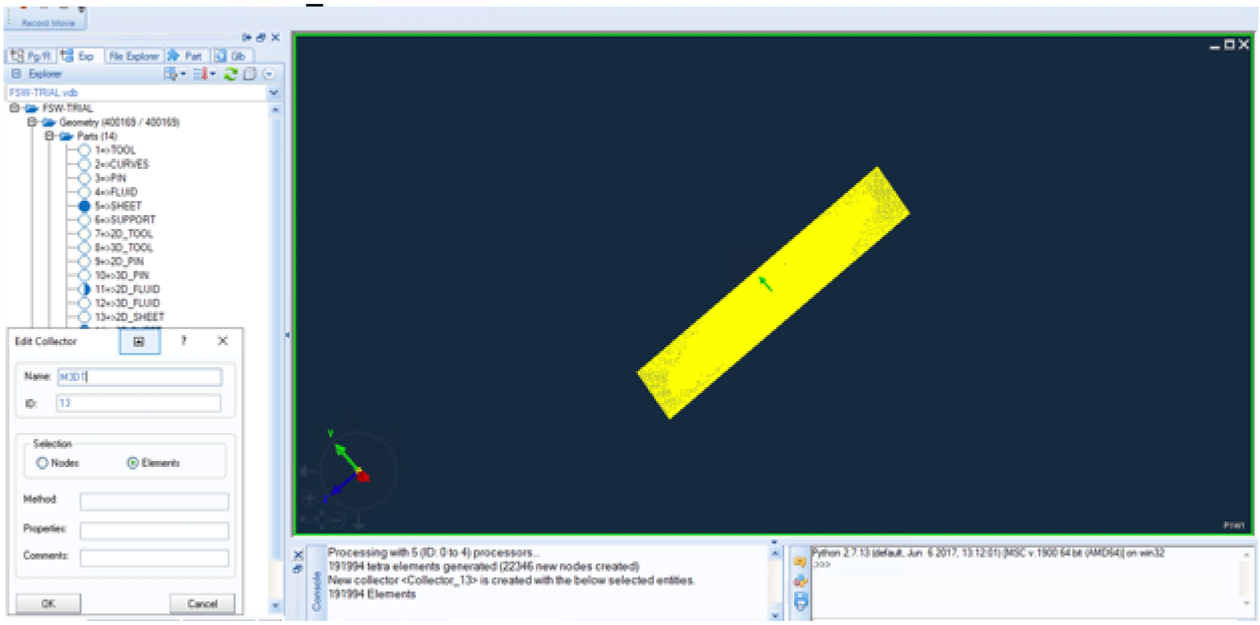


Figure 4.1.85 M3DT

49. Select **top surface and 2 side surfaces** which are parallel to X axis
50. **Add to New Collectors**
51. Rename **Collector_14** as **M3DT_AIR**

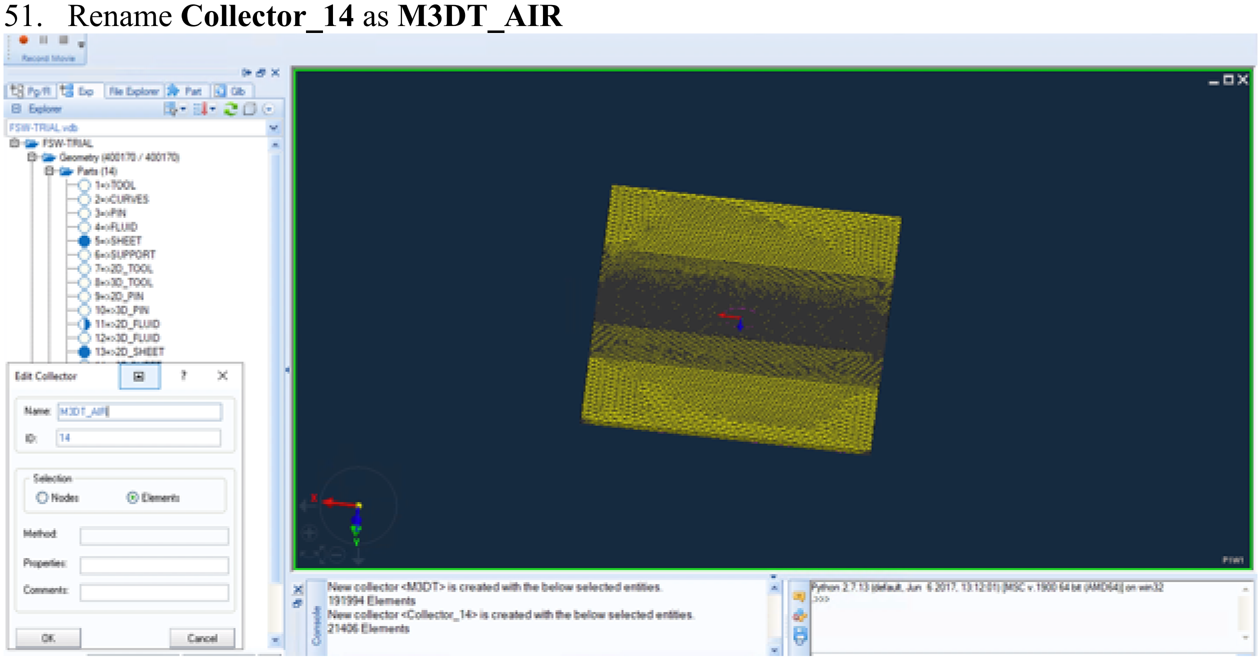


Figure 4.1.86 M3DT_AIR

Mesh SUPPORT

1. Choose **Tools**, click **Transform**
2. Select **bottom surface of SHEET**, choose **Copy**, **Part 15**
3. Click **Copy**

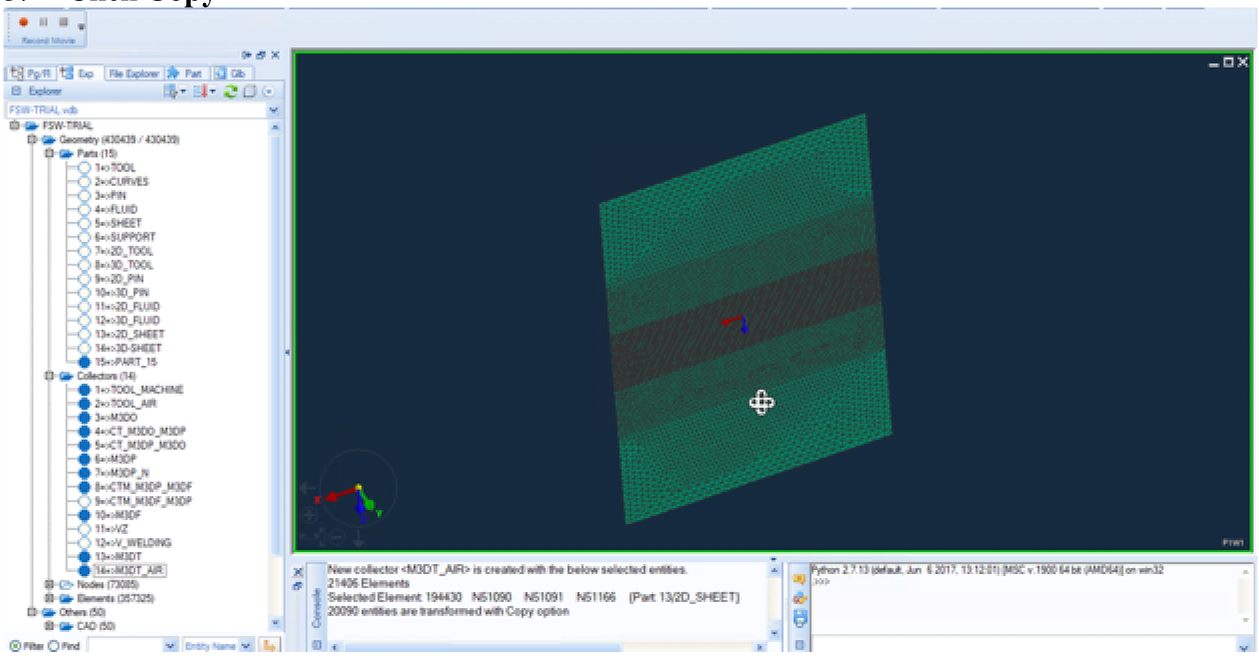


Figure 4.1.87 Copy surface

4. Choose **Checks**, click **Element Normals (Orientation)**
5. Choose **Arrows**, click **Flip Normals**, choose **Elements**, select **all elements in PART_15**
6. Click Apply

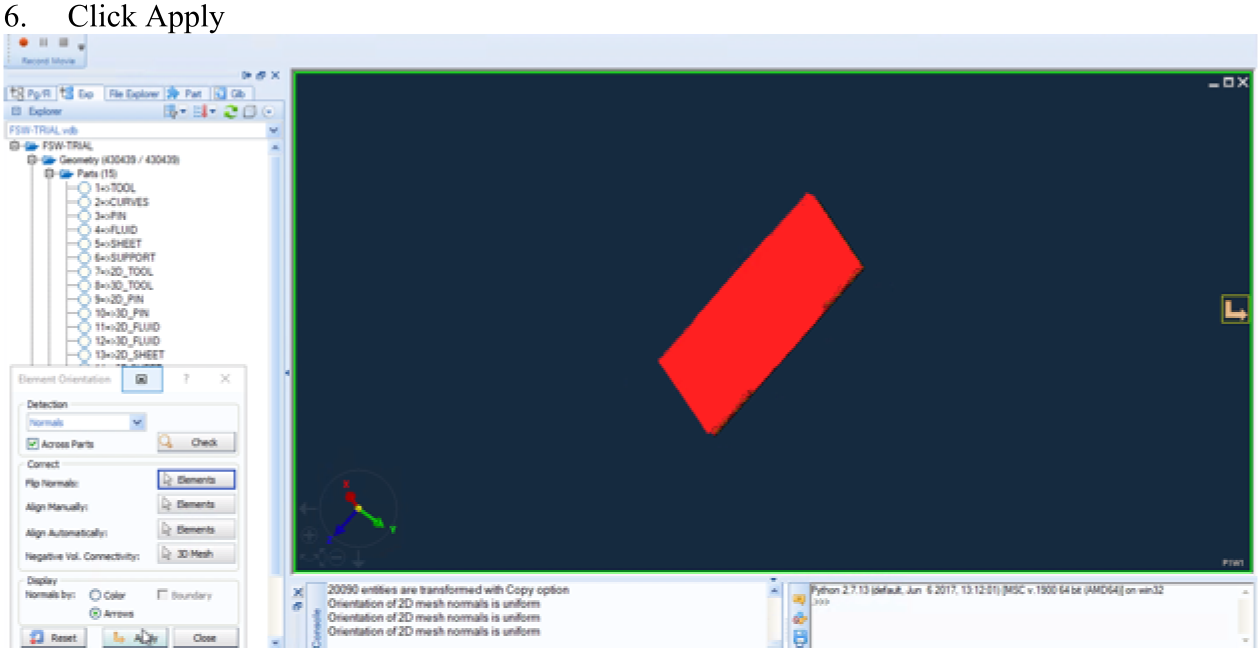


Figure 4.1.88 Flip Normals

7. Show **PART_15** and **SUPPORT**
8. Choose **2D**, click **Automesh Surface**, choose Front Surface
9. In **2D Mesh** window, **Set Element Size 15**, **Part 15**, reduce edge nodes from **10 to 6**, **20 to 15**
10. Click **OK**

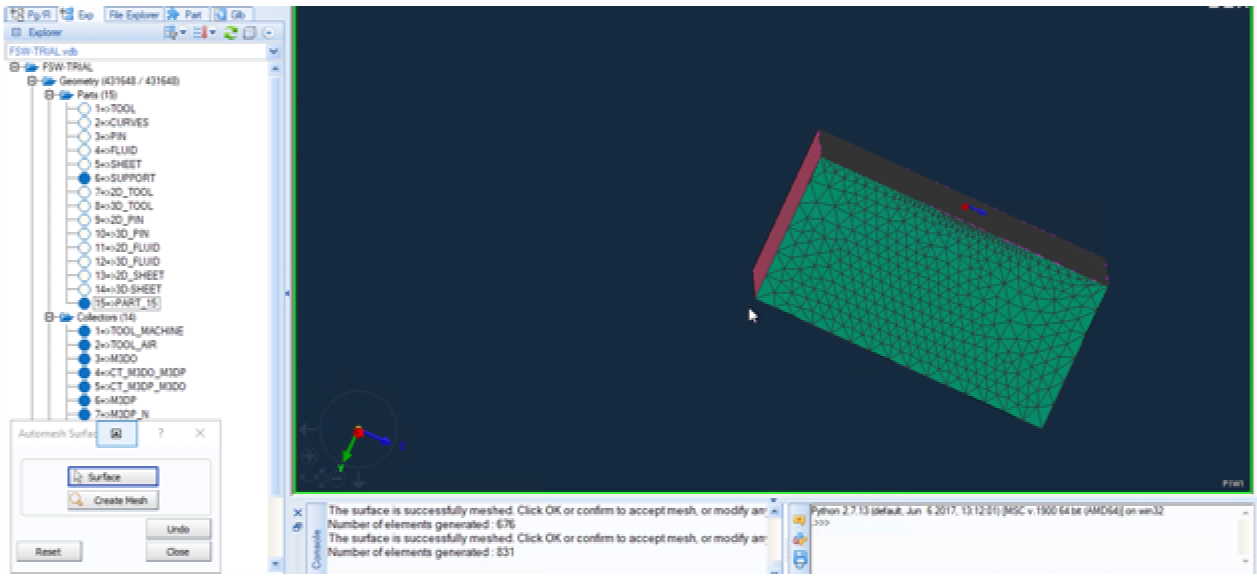


Figure 4.1.89 2D Mesh

11. Select 2 side surfaces, **Set Element Size 15**, reduce edge nodes from **10 to 6**, **13 to 11**
12. Click **OK**
13. Select back surface, **Set Element Size 15**
14. Click **OK**
15. Select bottom surface
16. Click **OK**

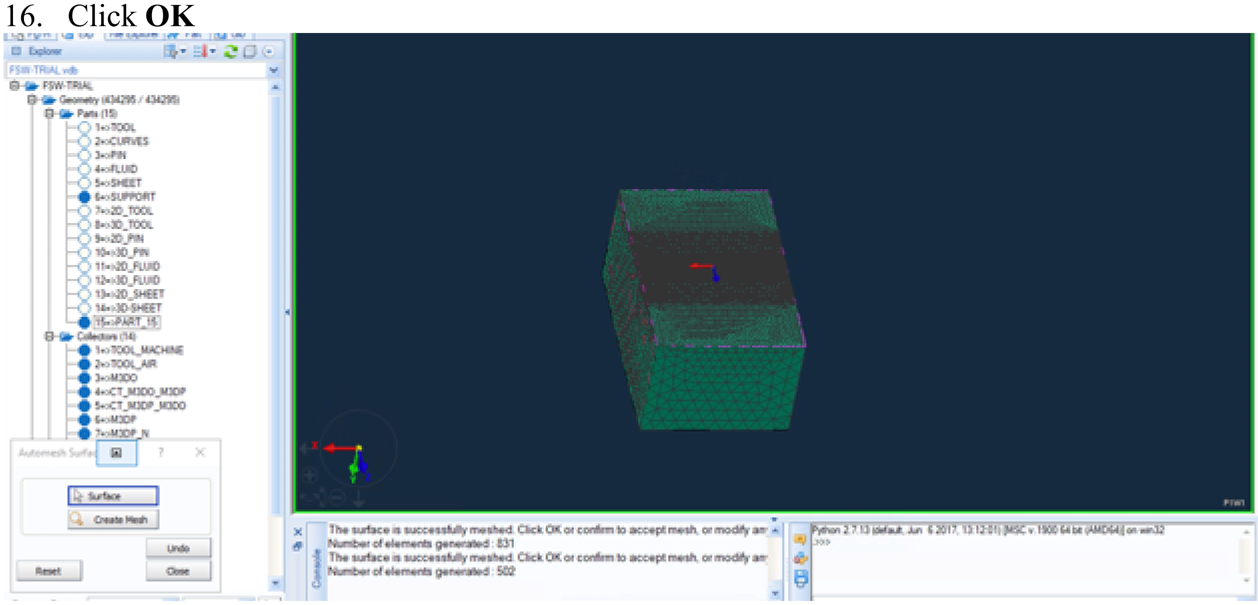


Figure 4.1.90 Part_15

17. Choose **3D**, click **Tetra Mesh**
18. Select all elements in **Part_15**, **Transition Factor 3.5**
19. Click **Apply**

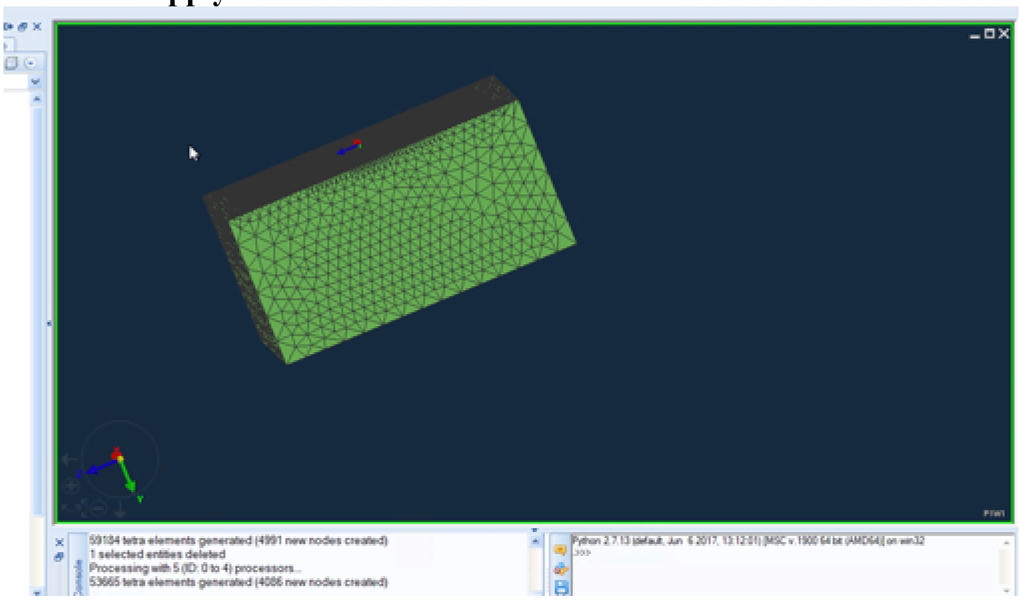


Figure 4.1.91 Part_16

20. Rename **PART_15** as **2D_SUPPORT**
21. Rename **PART_16** as **3D_SUPPORT**

Collectors 15-

1. Select the **bottom surface of 2D_SHEET**, **Add to New Collector**
2. Rename **Collector_15** as **CT_M3DT_M3DS**
3. Select the **top surface of 2D_SUPPORT**, **Add to New Collector**
4. Rename **Collector_16** as **CT_M3DS_M3DT**

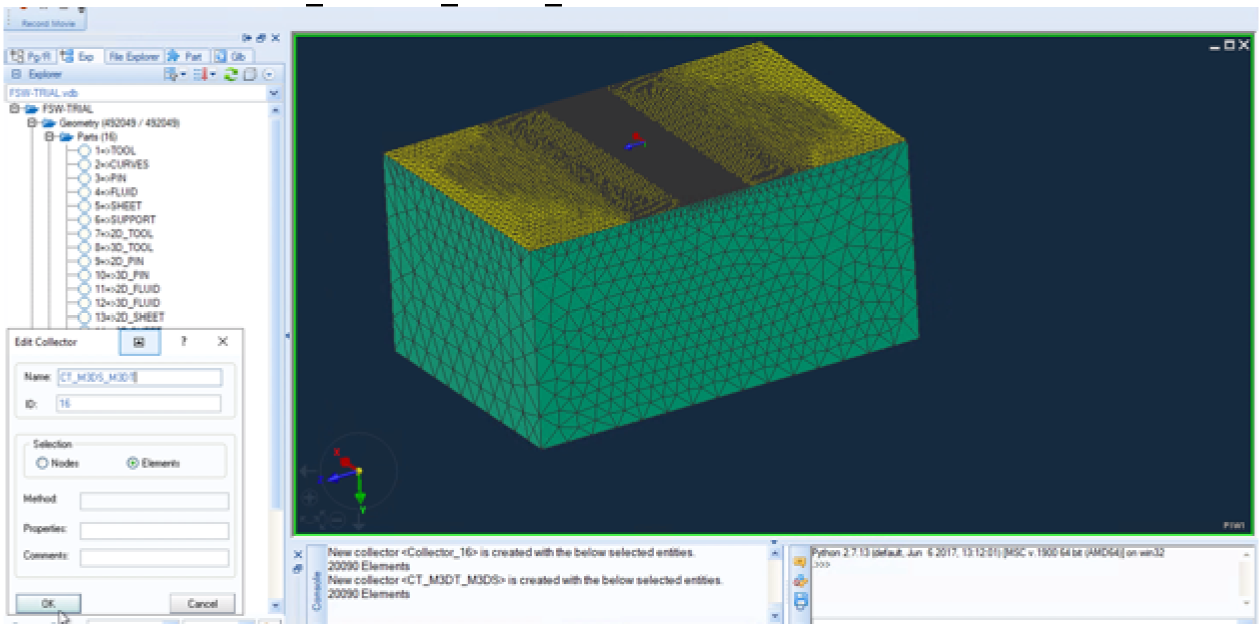


Figure 4.1.92 CT_M3DS_M3DT

5. Select **2 side surfaces which are parallel to X axis and bottom surface of SUPPORT**
6. **Add to New Collector,** Rename **Collector_17** as **M3DS_AIR**

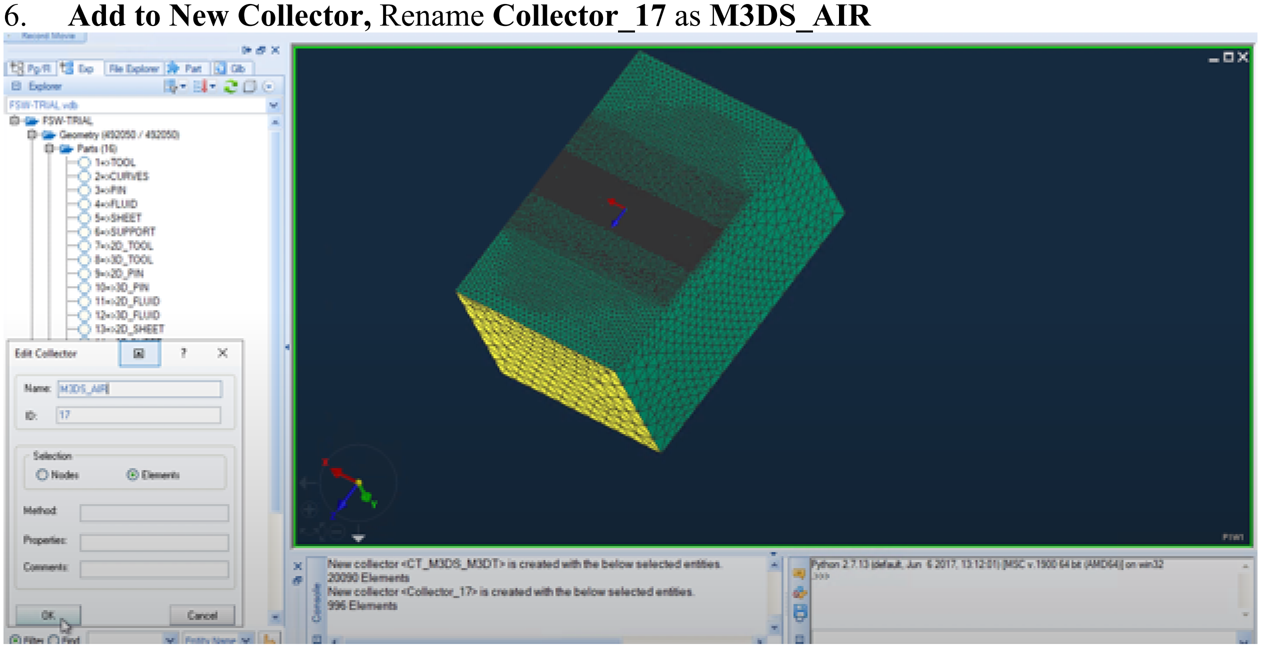


Figure 4.1.93 M3DS_AIR

7. Select **all elements in 3D_SUPPORT**, **Add to New Collector**
8. Rename **Collector_18** as **M3DS**
9. Show **3D_TOOL** and **3D_PIN**
10. In **Selection** window, choose **Node**.
11. Select **all nodes in the figure**, **Add to New Collector**
12. Rename **Collector_19** as **N_ROT**

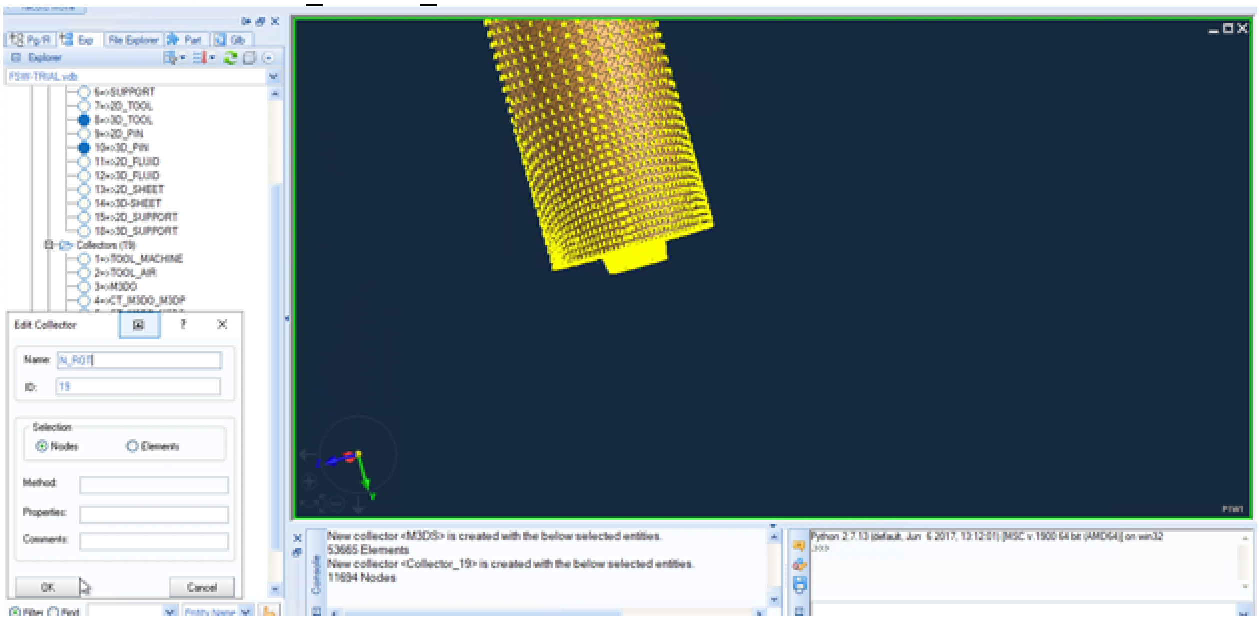


Figure 4.1.94 N_ROT

13. Show **3D_TOOL, 3D_PIN, 3D_FLUID, 3D_SHEET** and **3D_SUPPORT**
14. In **Views** window, choose **Bottom** View. **Selection: Node**
15. Select **all the front nodes** in X axis positive direction, **Add to New Collector**

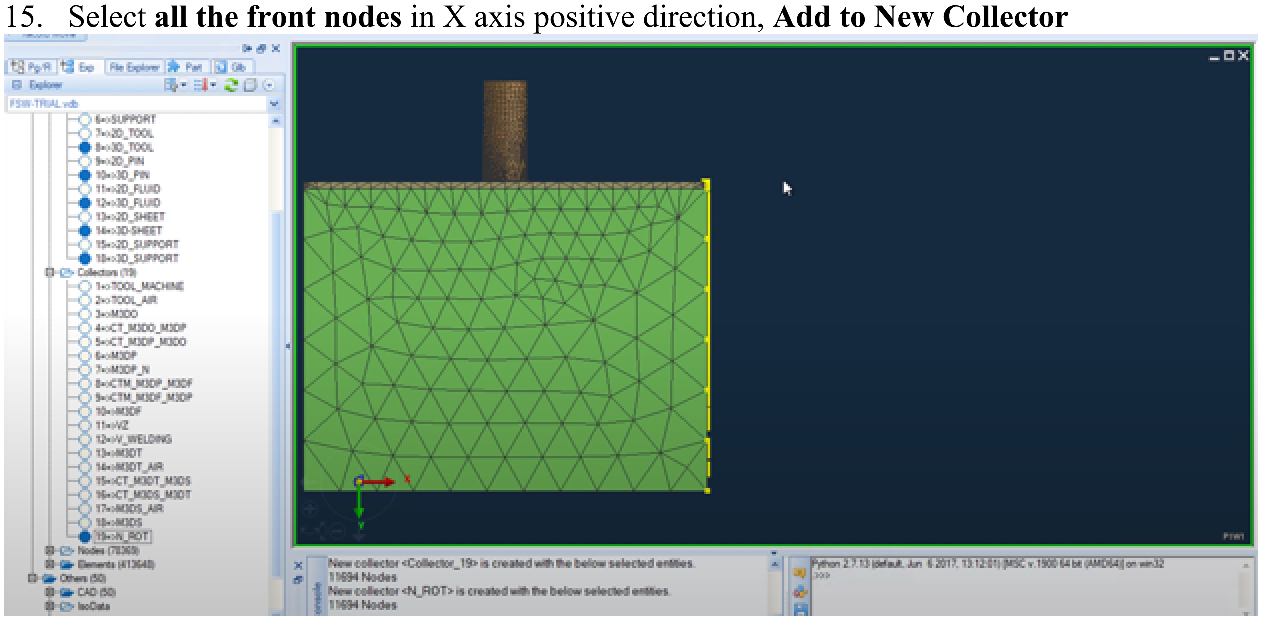


Figure 4.1.95 Collector_20

16. Rename **Collector_20** as **N_FRONT**
17. Show only **2D_PIN** and **2D_FLUID without bottom surface or side surface**

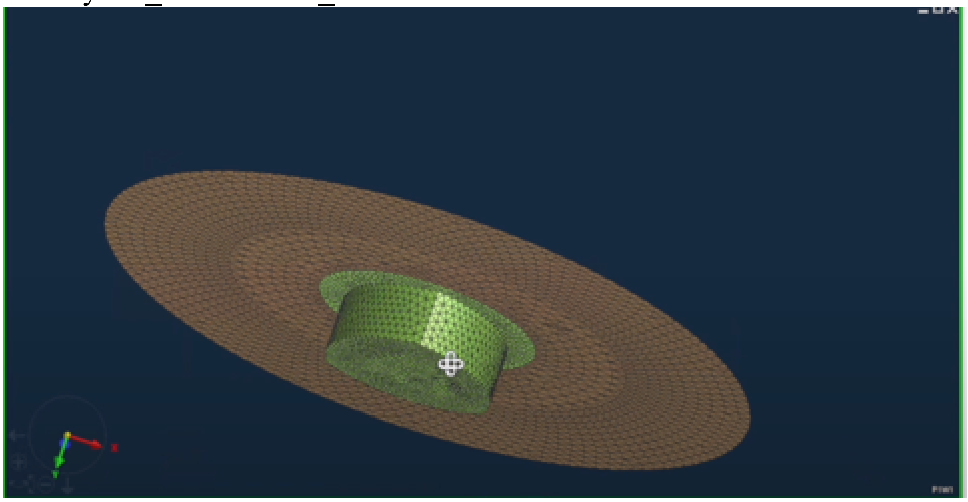


Figure 4.1.96 Part

18. Show **TOOL**, choose **Left-Side view**
19. Delete all the **visible 2D elements**

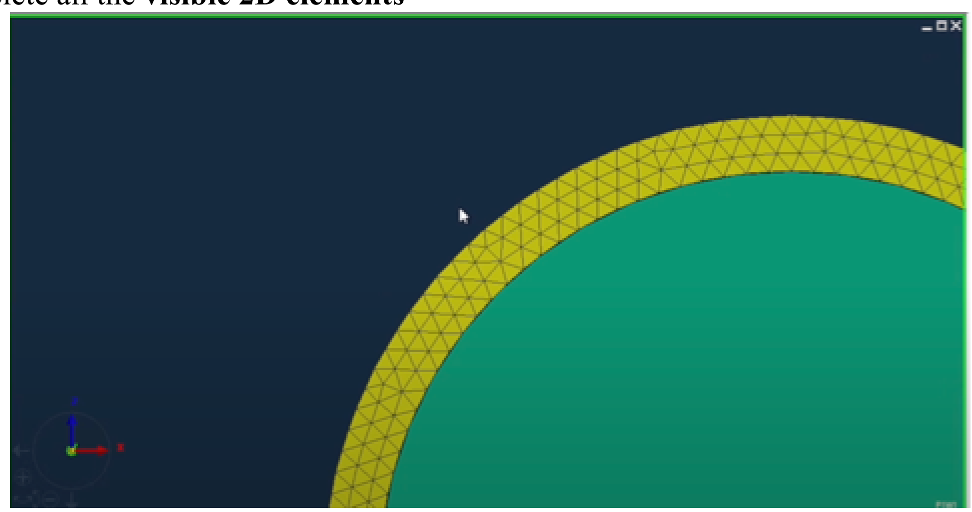


Figure 4.1.97 Delete part

20. **Selection: Node**, select **nodes of all the remaining part, Add to New Collector**
21. Rename **Collector_21** as **V_TOOL**

22. Select a tip node of the cutter edge, **Add to New Collector**
23. Rename **Collector_21** as **NODE_STAB**

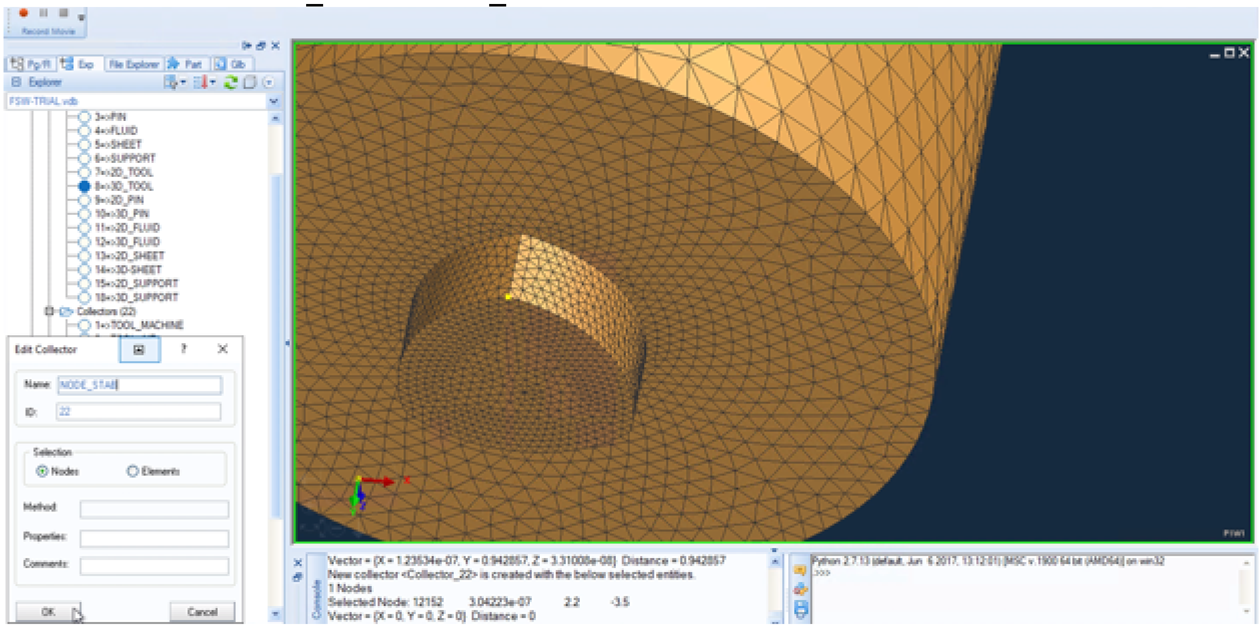


Figure 4.1.72 NODE_STAB

24. Select all. Choose **Tools**, click **Transform**
25. In **Transform** window, choose **Rotate**,input **Angle 90**, choose **Move**
26. Click **Move**

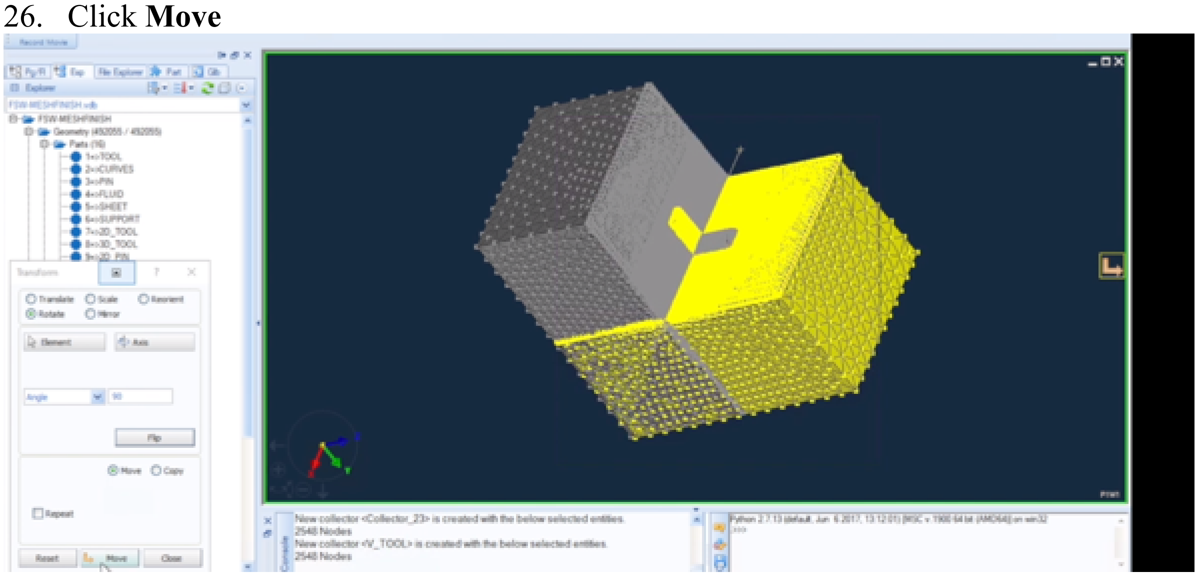


Figure 4.1.73 Rotate Part

4.2 Simulation execution

4.2.1 Export the model

From the **File** menu, choose **Export**

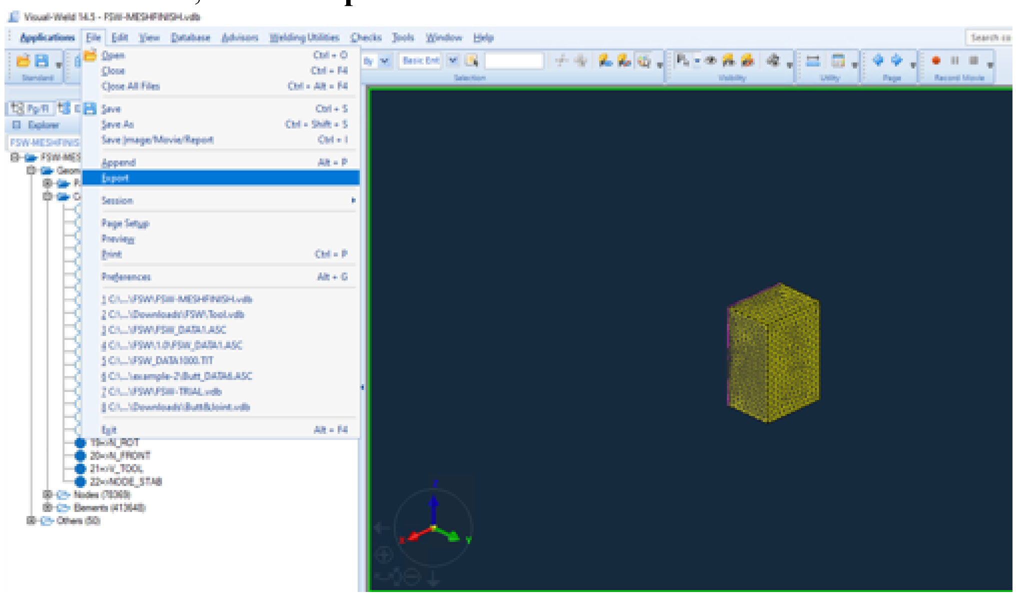


Figure 4.2.1 Export the model

EXPORT

1. In the **Export** window, choose **Files of type** as **SYSWORLD ASC data files**

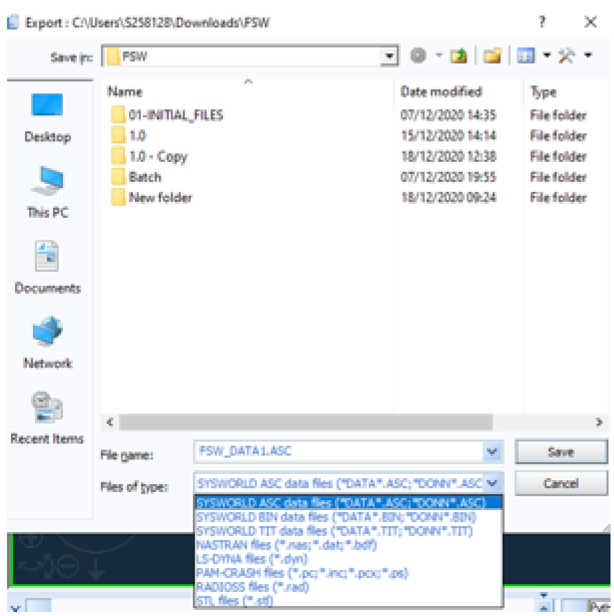


Figure 4.2.2 Save as .ASC file

2. Give the **File name** as **FSW_DATA1**(as an example)
3. Click **Save**

4.2.2 **Files assemble**

Build a **New Folder** in your working directory

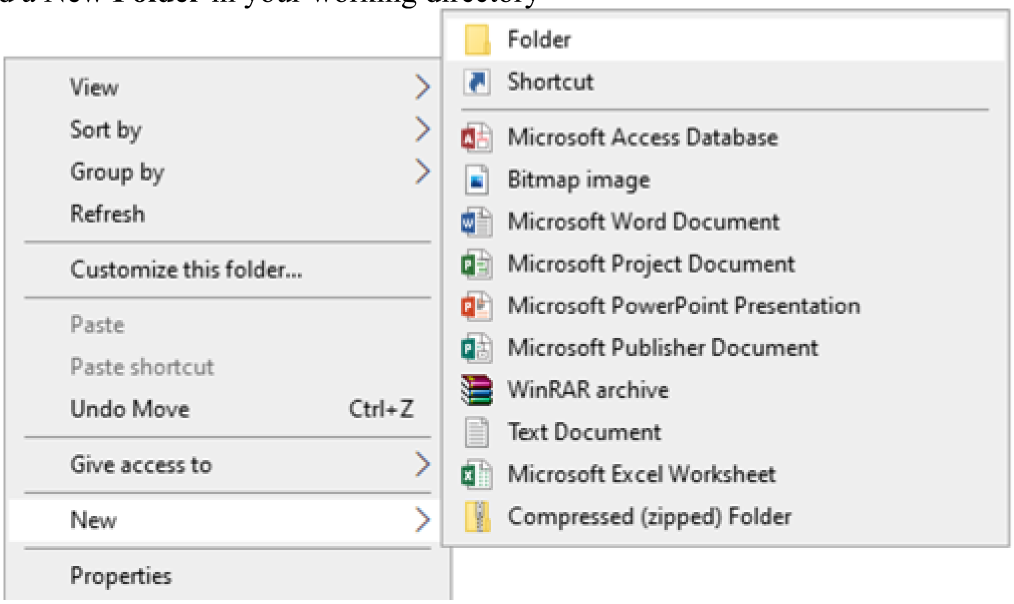


Figure 4.2.3 New folder

Find and **Click** the folder **01-INITIAL_FILES** inside the startup directory

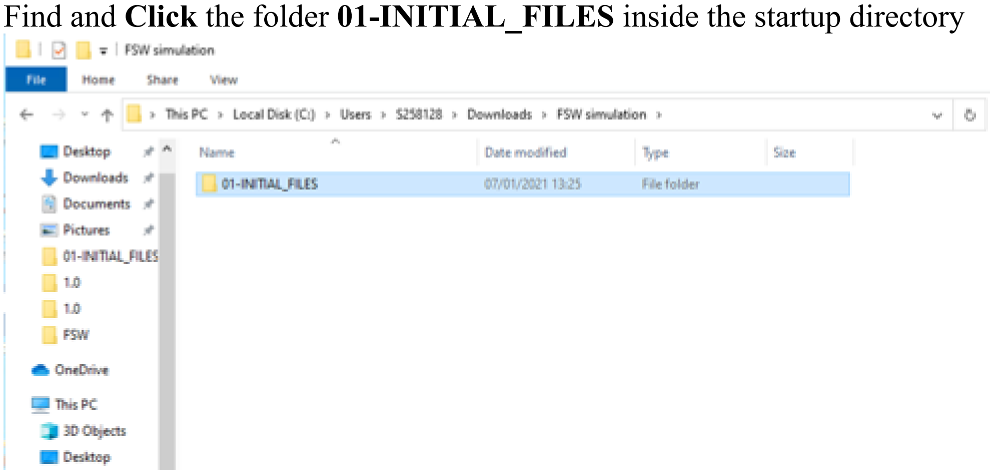


Figure 4.2.4 01-INITIAL_FILES

Copy the files with name **FSW_01**, **LAUNCH**, **MAT**, **parameters**, **post_tnl.par** and **thermocouples**

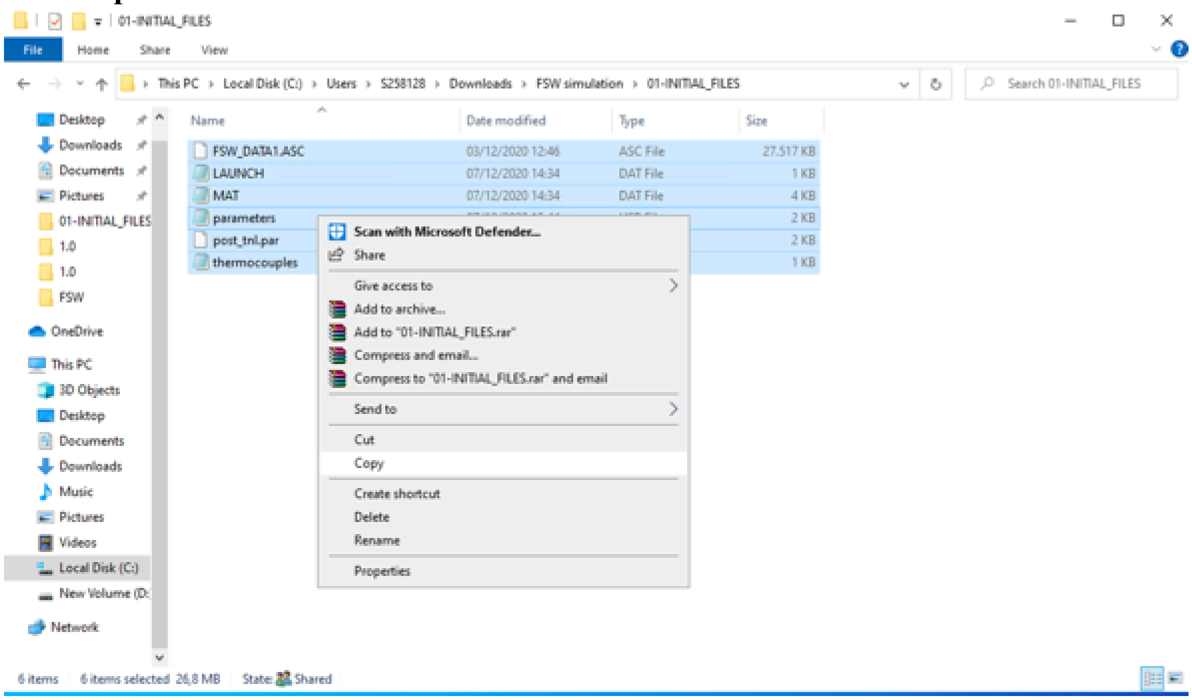


Figure 4.2.5 Copy 6 files

Paste the files in your **New Folder**

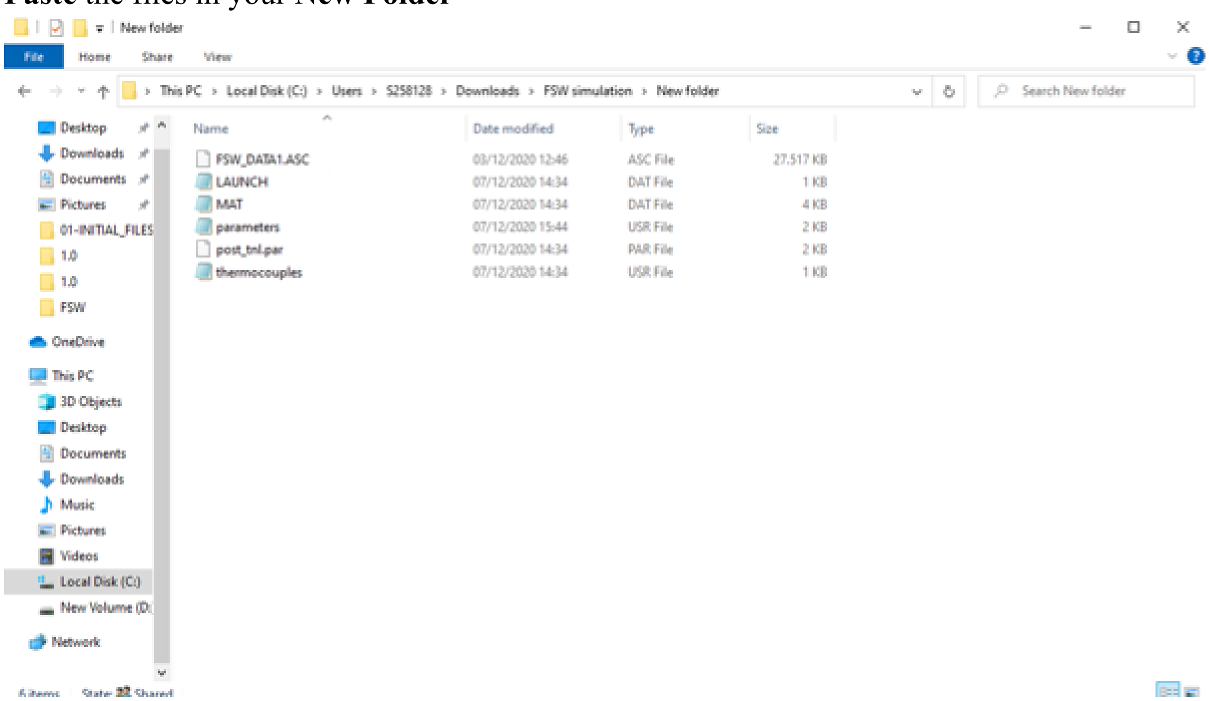


Figure 4.2.6 Paste files in New folder

4.2.3 **Parameter setup**

Open the file **parameters**

```
parameters - Notepad
File  Edit  Format  View  Help
; Parameter file for FSW computation

;***************************** Model ************************
TITL (title of the model)                              > FSW

;*************** Mesh ***************
ROOT (root name of the mesh)                           > FSW_
MESH (number of the ascii mesh)                        > 1

;***************************** Process **********************
AVANce (weld speed mm/s)                               > 3.0
ROTAtion (rotation speed rad/s)                        > 68.067
TEMP                                                   > 20.0
CONFIG                                                 > 1

;***************************** Computation ******************
INCREment per round                                    > 33
STOP (critera)                                         > 2
DELTa Temperature                                      > 5
REVO                                                   > 1
;STATionnary point                                    >
MATE (material properties)                             > MAT.DAT
REST (restart)                                         > 0

;***************************** Post Processing **************
DEPO (type of post-processing                          > 2
MINIERF                                                > 2
;DTPOints                                             > thermocouples.usr

;***************************** Convergence Management *******
Ln 1, Col 1    100%    Unix (LF)    UTF-8
```

Figure 4.2.7 parameters in notepad

Input values shown in Table 4.2.8

Table 4.2.8 Parameters

Name	Value	Meaning
TITL	FSW	The type of model is friction stir welding
ROOT	FSW_	Root name of the mesh
MESH	1	Number of the ascii mesh (from example FSW_01)
AVANce	3.0	Weld speed mm/s
ROTAtion	68.067	Rotation speed rad/s
TEMP	20	External temperature
CONFIG	1	Joint type, 1 Quare butt joint; 2 T joint; 3 Lap joint, see Figure.1.3.2
INCREment per round	33	Number of triangles based on your mesh, see Figure.1.3.3
STOP	2	To stop simulation 1 number of rounds; 2 temperature increment
DELTa Temperature	5	Temperature increment
MATE	MAT.DAT	File name for material properties storage.

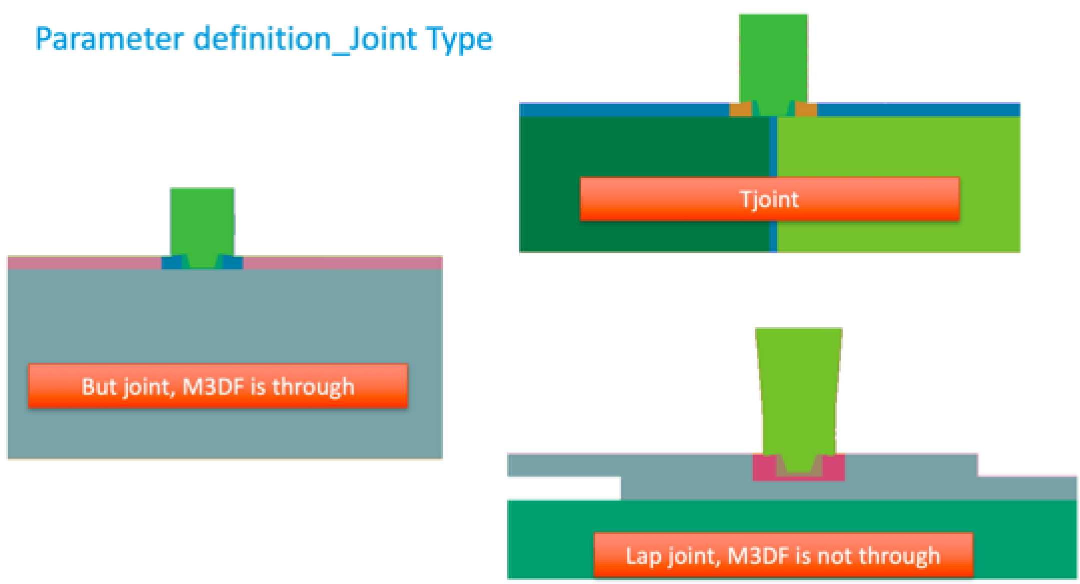


Figure 4.2.9 Joint Type

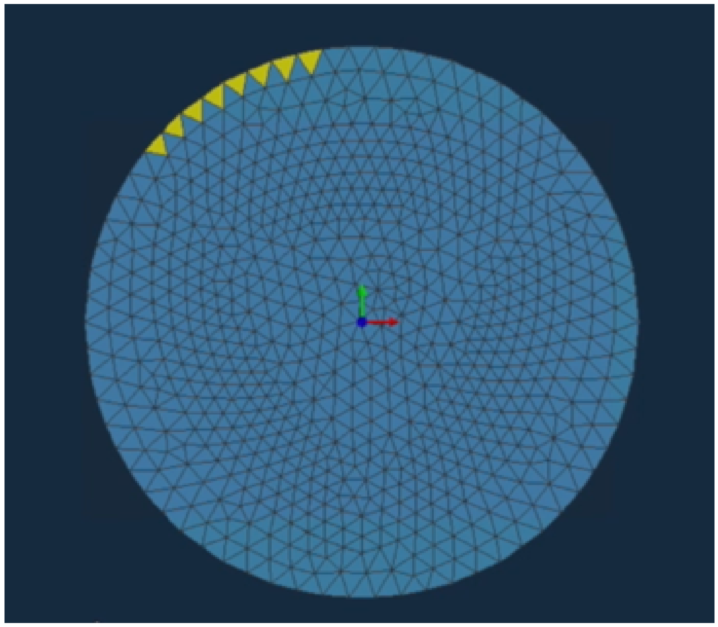


Figure 4.2.10 Edge of pin part

The rest of parameters which have not mentioned will remain unchanged.
Save and **Close**

Figure 4.2.11 Save the data

4.2.4 Material Properties setup

The material properties can be discovered based on your experimental data, online library or database from Sysweld software. This instruction only introduce how to query from the database of Sysweld software.

Material Database

1. From **Databse** menu, click on **Material**

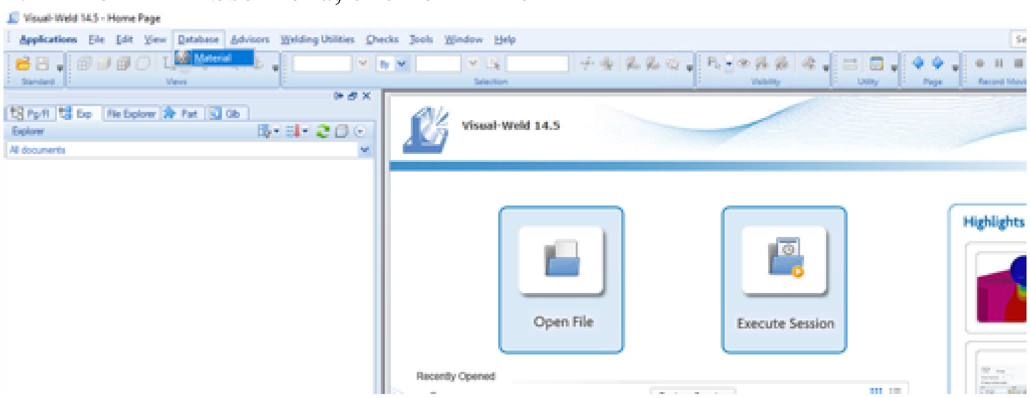


Figure 4.2.12 Material Database

2. In **Weld** folder, click **5000 Series Al Alloy**

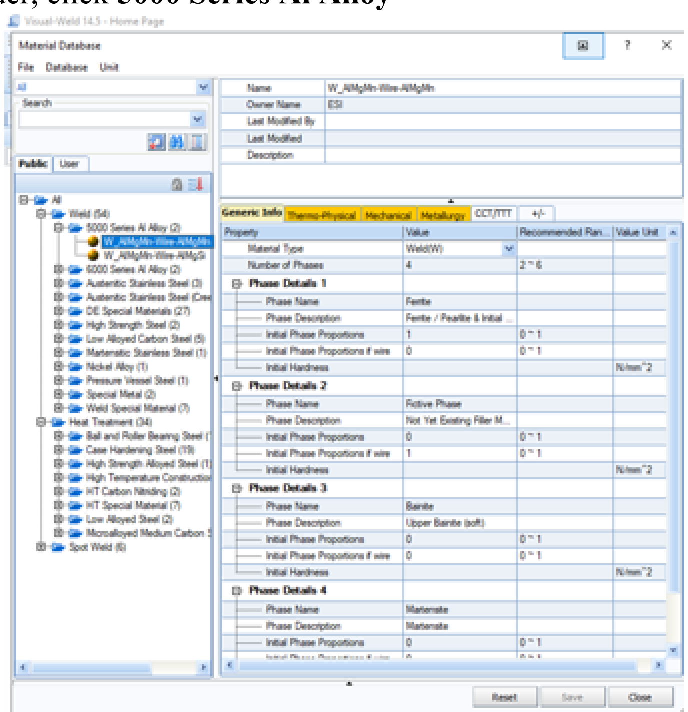


Figure 4.2.13 Al Alloy

3. Choose **W_AlMgM-Wire-AlMgMn** material, click on **Thermo-Physical** or **Mechanical**. There are a lot types of properties data available. We can see their specific value under different temperatures

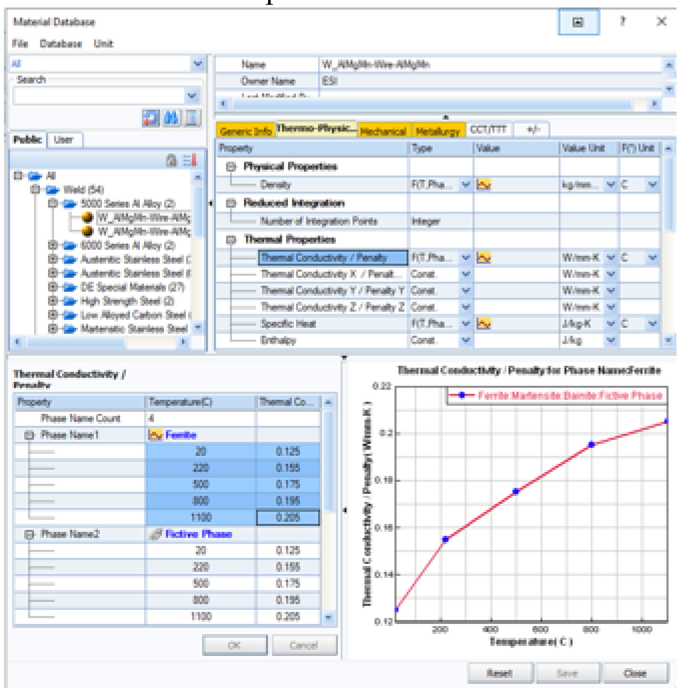
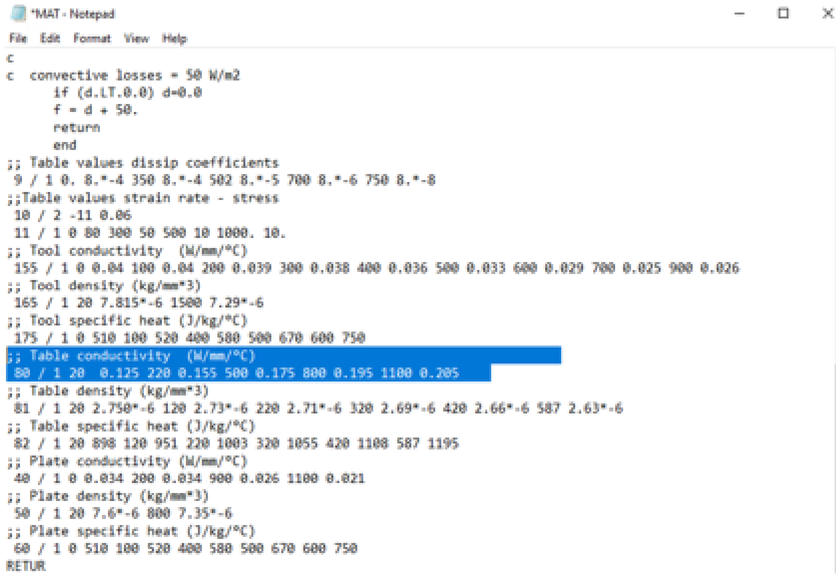


Figure 4.2.8 Thermal Conductivity

Data Input

1. **Open** the file **MAT.DAT**
2. Under the line of **Table Conductivity**, input **80 / 1 20 0.125 220 0.155 500 0.175 800 0.195 1100 0.205**



```
*MAT - Notepad
File  Edit  Format  View  Help
c
c  convective losses = 50 W/m2
      if (d.LT.0.0) d=0.0
      f = d + 50.
      return
      end
;; Table values dissip coefficients
 9 / 1 0. 8.*-4 350 8.*-4 502 8.*-5 700 8.*-6 750 8.*-8
;;Table values strain rate - stress
 10 / 2 -11 0.06
 11 / 1 0 80 300 50 500 10 1000. 10.
;; Tool conductivity  (W/mm/°C)
 155 / 1 0 0.04 100 0.04 200 0.039 300 0.038 400 0.036 500 0.033 600 0.029 700 0.025 900 0.026
;; Tool density (kg/mm*3)
 165 / 1 20 7.815*-6 1500 7.29*-6
;; Tool specific heat (J/kg/°C)
 175 / 1 0 510 100 520 400 580 500 670 600 750
;; Table conductivity  (W/mm/°C)
 80 / 1 20  0.125 220 0.155 500 0.175 800 0.195 1100 0.205
;; Table density (kg/mm*3)
 81 / 1 20 2.750*-6 120 2.73*-6 220 2.71*-6 320 2.69*-6 420 2.66*-6 587 2.63*-6
;; Table specific heat (J/kg/°C)
 82 / 1 20 898 120 951 220 1003 320 1055 420 1108 587 1195
;; Plate conductivity (W/mm/°C)
 40 / 1 0 0.034 200 0.034 900 0.026 1100 0.021
;; Plate density (kg/mm*3)
 50 / 1 20 7.6*-6 800 7.35*-6
;; Plate specific heat (J/kg/°C)
 60 / 1 0 510 100 520 400 580 500 670 600 750
RETUR
```

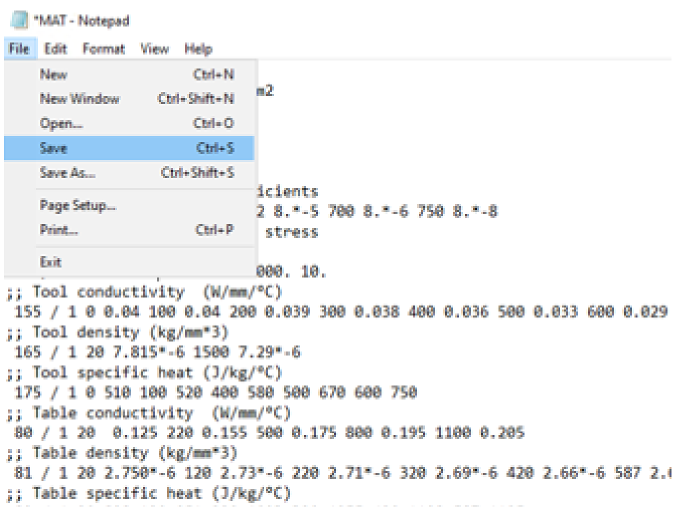
Figure 4.2.15 Input the Thermal Conductivity

The numbers mean:

80	means the code number for table conductivity
1	means the data is composed of 1 line
20 0.125	means under 20 °C, the conductivity is 0.125 W/mm/°C
220 0.155	means under 220 °C, the conductivity is 0.155 W/mm/°C

etc.

We need to modify the values of **conductivity**, **density**, **specific heat**, **strain rate - stress** and **dissipation coefficients**.
By input all the data manually, we assign each properties of our material to the model.
Save and **Close**



```
*MAT - Notepad
File  Edit  Format  View  Help
New                Ctrl+N
New Window   Ctrl+Shift+N     m2
Open...            Ctrl+O
Save               Ctrl+S
Save As...    Ctrl+Shift+S
                               icients
Page Setup...                 2 8.*-5 700 8.*-6 750 8.*-8
Print...           Ctrl+P      stress
Exit                          000. 10.
;; Tool conductivity  (W/mm/°C)
 155 / 1 0 0.04 100 0.04 200 0.039 300 0.038 400 0.036 500 0.033 600 0.029
;; Tool density (kg/mm*3)
 165 / 1 20 7.815*-6 1500 7.29*-6
;; Tool specific heat (J/kg/°C)
 175 / 1 0 510 100 520 400 580 500 670 600 750
;; Table conductivity  (W/mm/°C)
 80 / 1 20  0.125 220 0.155 500 0.175 800 0.195 1100 0.205
;; Table density (kg/mm*3)
 81 / 1 20 2.750*-6 120 2.73*-6 220 2.71*-6 320 2.69*-6 420 2.66*-6 587 2.
;; Table specific heat (J/kg/°C)
 82 / 1 20 898 120 951 220 1003 320 1055 420 1108 587 1195
```

Figure 4.2.16 Save the material properties

4.2.5 Launch Simulation

Open the solver **SYSWELD (English) Solvers 2019.0** on your pc

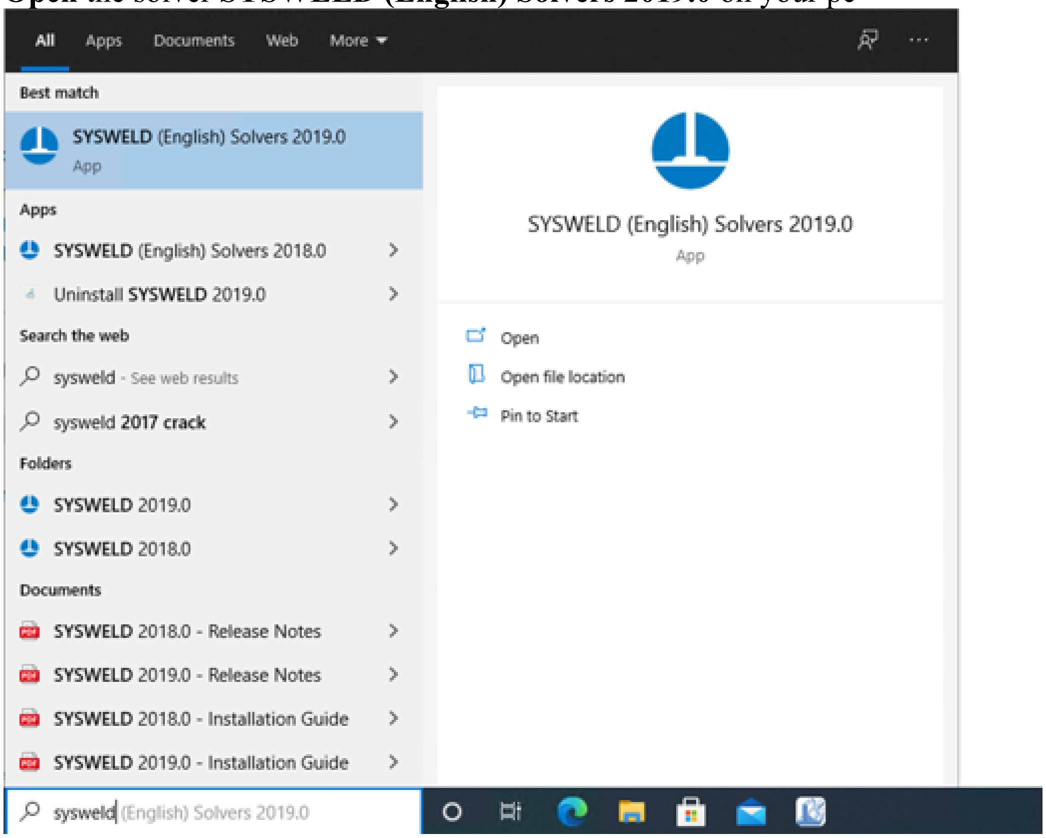


Figure 4.2.17 Open SYSWELD Solvers

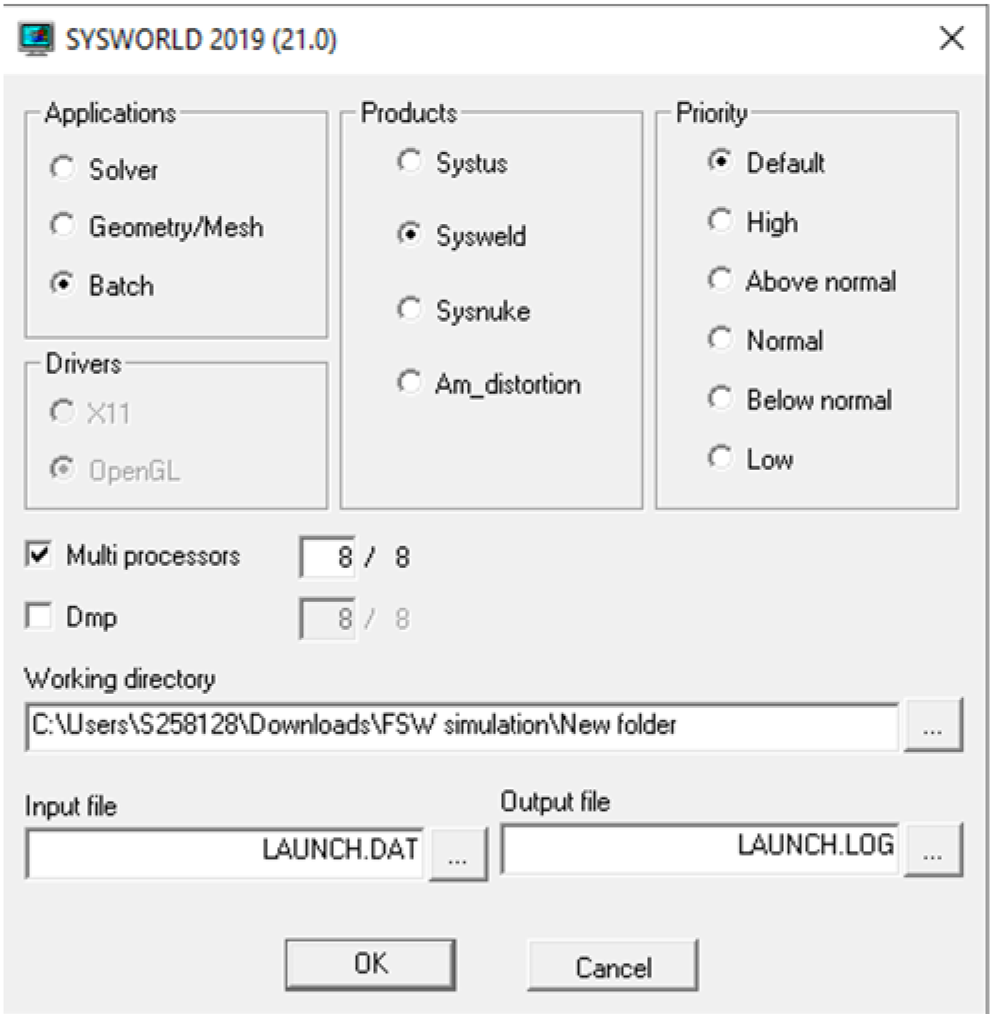


Figure 4.2.18 SYSWELD Solvers UI

SYSWELD Solvers

1. In **Applications**, choose **Batch**
2. In **Products**, choose **Sysweld**
3. In **Priority**, choose **Default**
4. Click **Multi processors**
5. In **Working directory**, choose **Your working directory\New Folder**
6. In **Input file**, choose **LAUNCH.DAT**
7. Click **OK**

The calculation of your model will start immediately.

4.3 Check results

Open the software **Visual-Viewer**

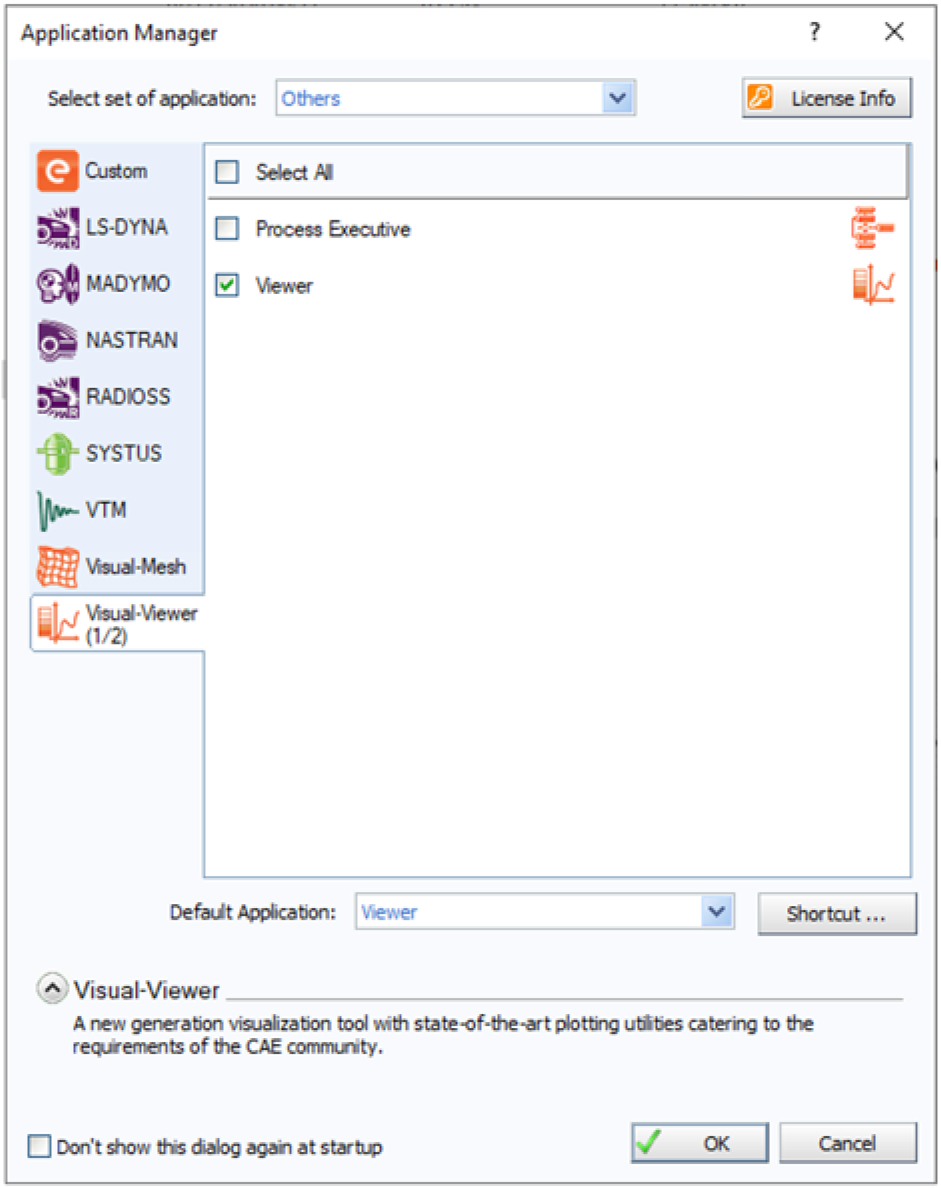


Figure 4.3.1 Open Visual-Viewer

Visual-Viewer

1. Click **Open File**, choose **Your working directory\New Folder\Butt_POST1000.erfh5**

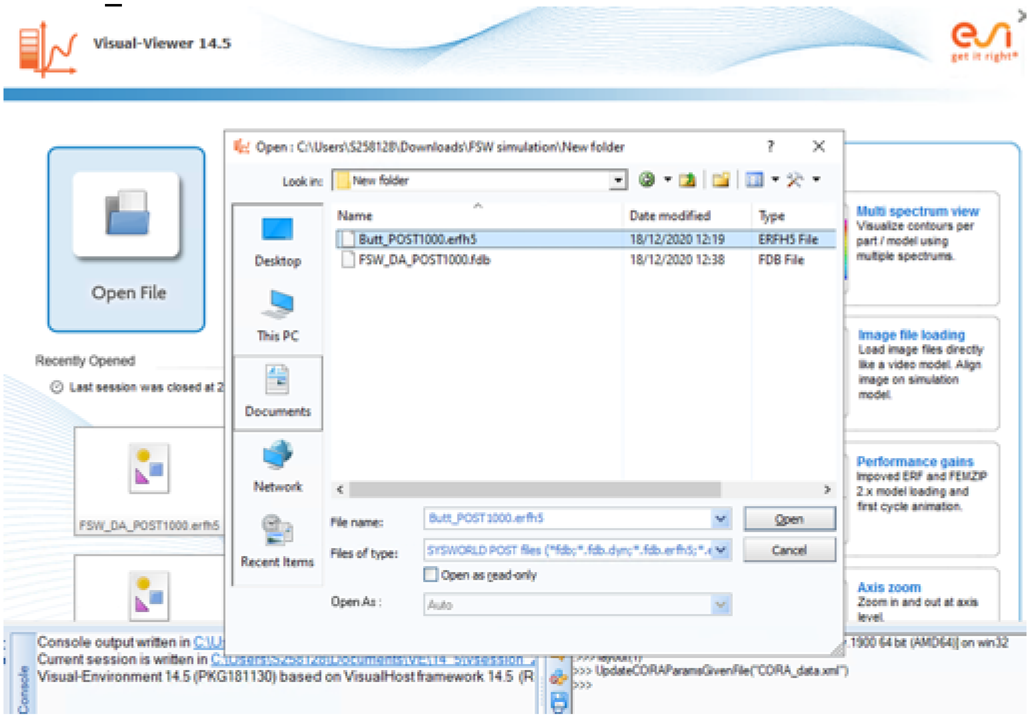


Figure 4.3.2 Open File Butt_POST1000.erfh5

2. Click **Results**
3. In the **Results** Window, click **Contour**

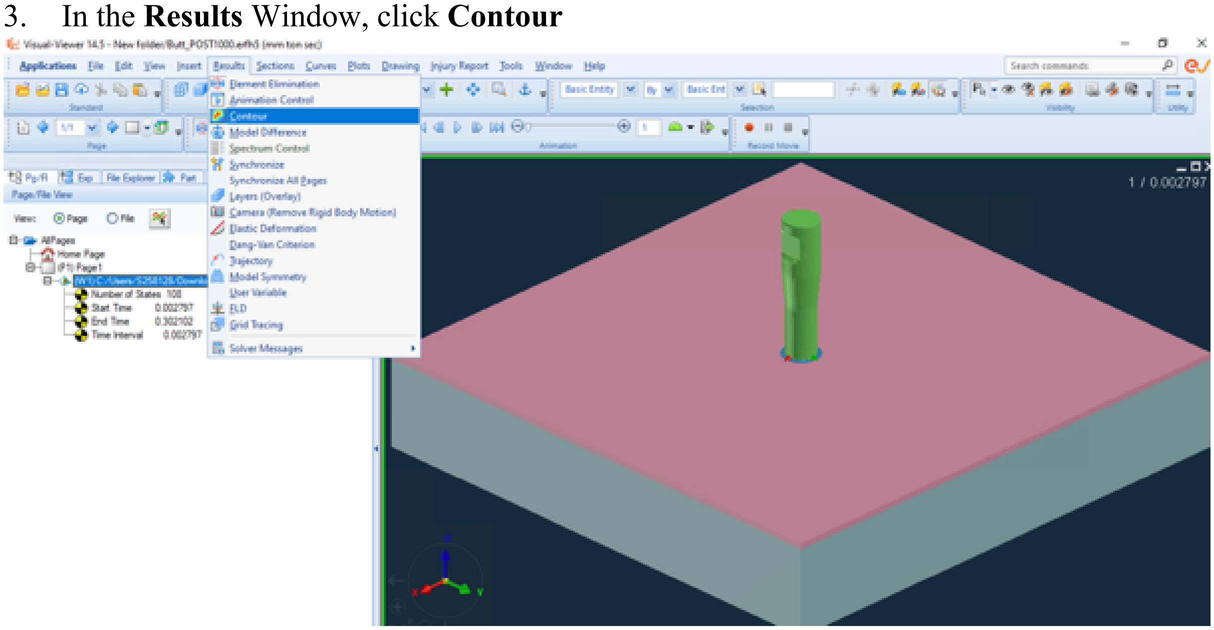


Figure 4.3.3 Check contour results

4. In the **Contour** Window, click **Temperature_NOD** or **VELOCITY_NOD** to see detailed temperature and velocity field.

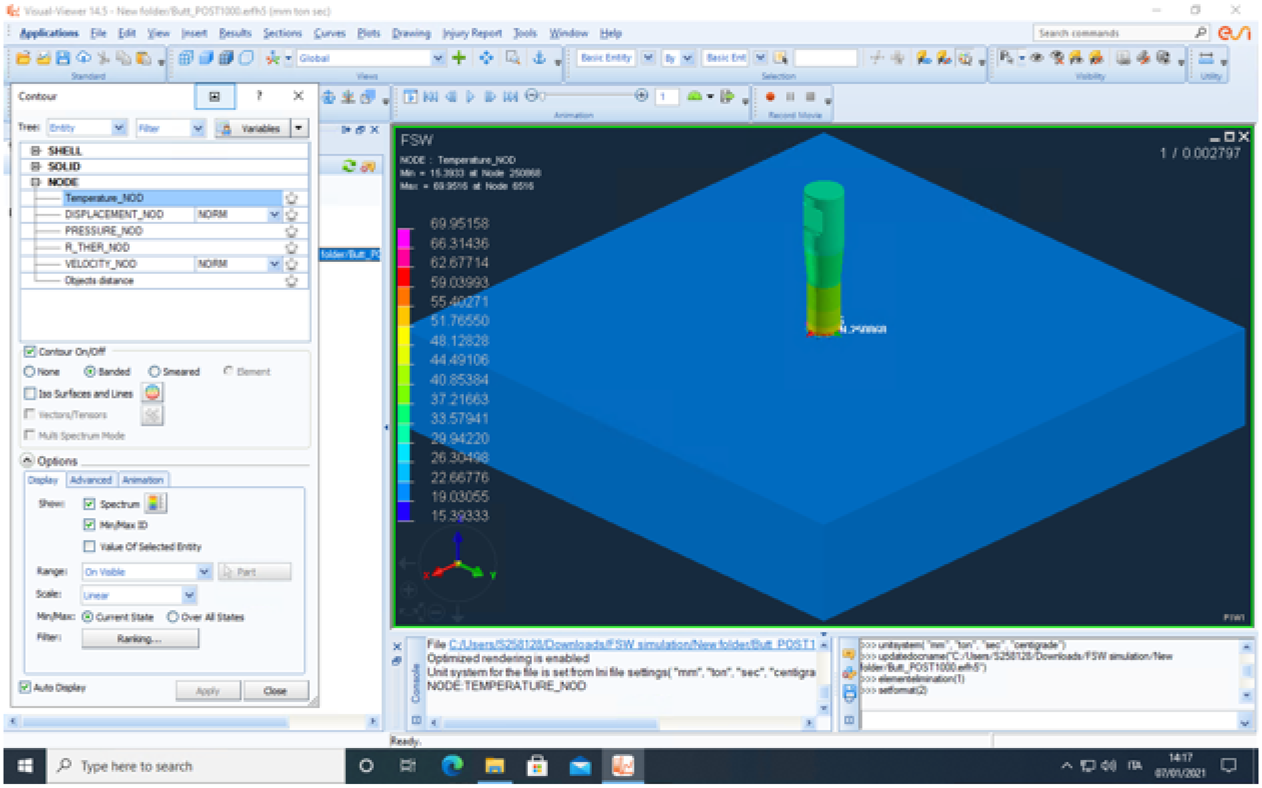


Figure 4.3.4 Check Nodes temperature

5 CHAPTER

RESULT ANALYSIS AND FUTURE DEVELOPMENTS

5.1 Experiment parameters

	Unit	1	2	3	4	5
Weld speed	[mm/min]	50	100	200	100	100
Rotation speed	[rpm]	1750	1750	1750	1250	2250
Ambient Temperature	[C°]	28	28	28	28	28
Vertical force	[kN]	1.7	1.8	2.15	2.1	1.5
Tmax at the sample surface close to the tool	[C°]	440	367	332	310	368

Table 5.1.1 Experiment parameters

As what we have mentioned on 4.2.4, we have carried out several different simulations to verify what we have obtained from our lab experiment. We simply modify the parameters which have been listed in Table 5.1.1, we can obtain our simulation results.

5.2 Materials

For our welding tool which is rotating above the weldment, we use the Tungsten/Rhenium alloy (75% W, 25% Re).

For our lap joint, we use a Aluminum alloy 6016 grade - DC05 steel (very low carbon steel). Aluminum sheet at the tool side, steel at the worktable side.

For the support which is used as a worktable, we use mateiral 39NiCrMo3.

5.3 Simulation results

In the figures of simulation results. The top half sheet is Aluminum alloy 6106 while the bottom half sheet is DC05 steel. Aluminium 6106 has a higher thermal conductivity and higher specific heat compared to DC05 steel.

Under the simulation conditions, the sheet is considered as infinite length along the welding direction. And this simulation process will stop only if the temperature variation is smaller than 5 degrees. So in this case, we have the same starting phase for all these 5 groups of experiment. Starting from the room temperture 28 degrees, we will only check the temperature field distribution in the end phase.

5.3.1 Weld speed 50mm/min, Rotation speed 1750 rpm

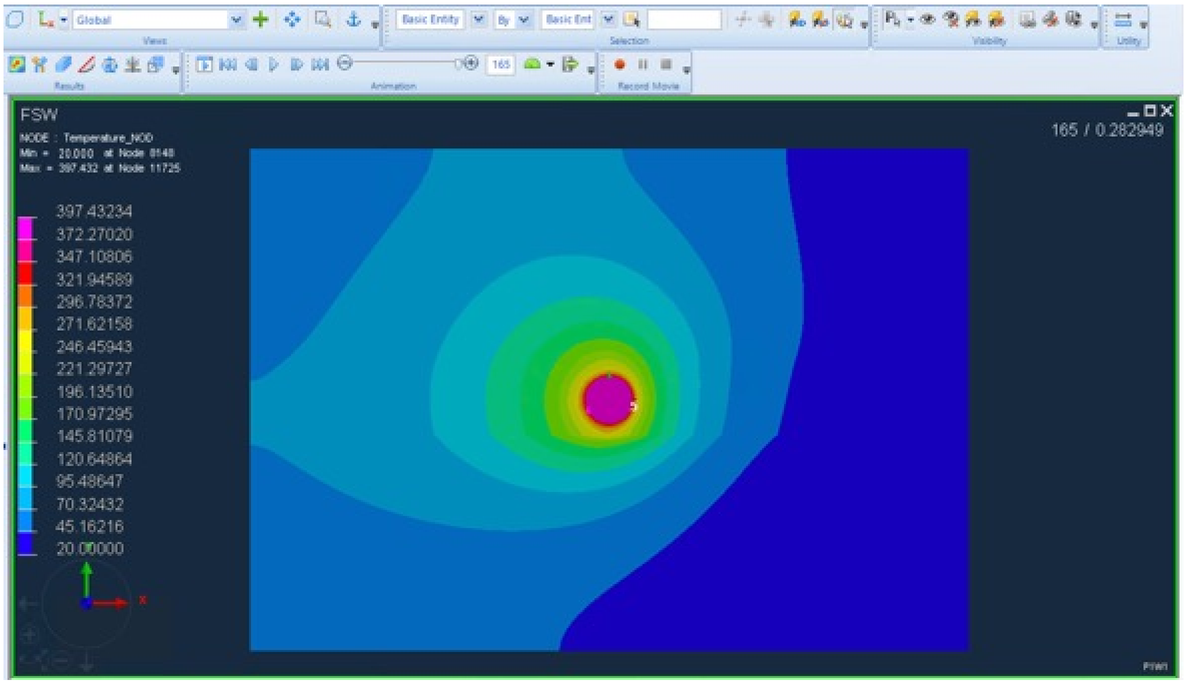


Figure 5.3.1 Temperature field of group 1

From the figure 5.3.1, we can see that the maximum temperature is in the node 11725 at tool edge. And the maximum value of temperature is 397.4 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side. The simulation takes 165 steps to reach this end phase.

5.3.2 Weld speed 100mm/min, Rotation speed 1750 rpm

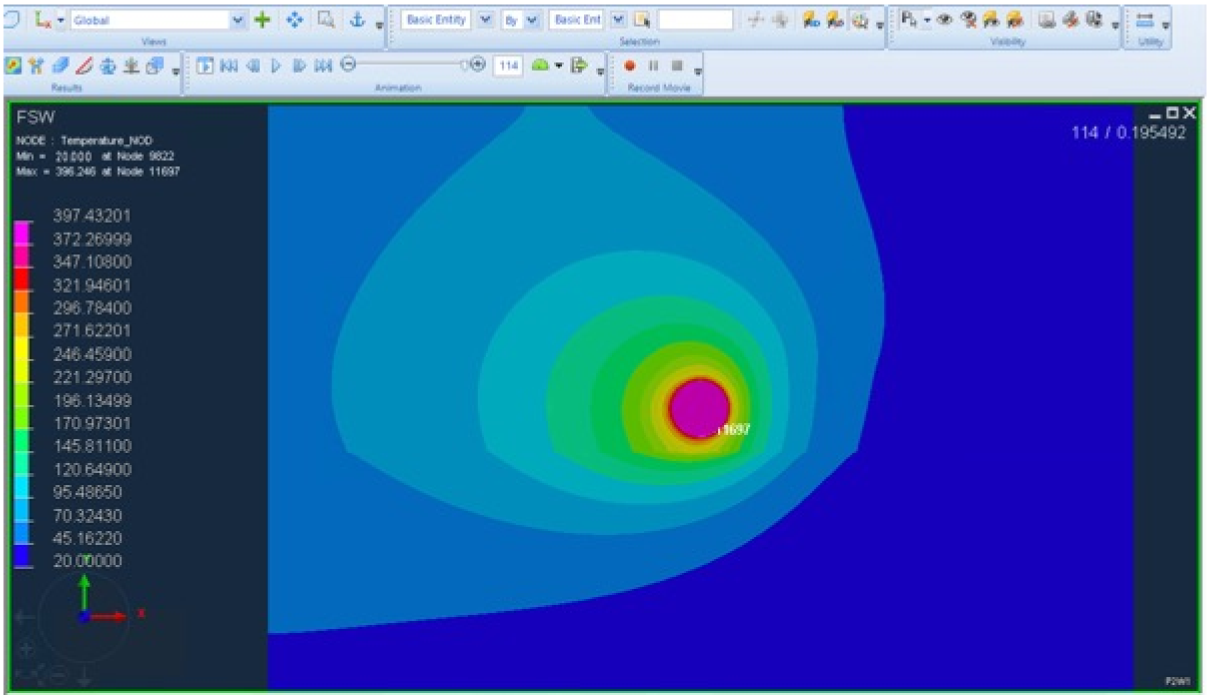


Figure 5.3.2 Temperature field of group 2

From the figure 5.3.2, we can see that the maximum temperature is in the node 11697 at tool edge. And the maximum value of temperature is 396.2 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 1, the maximum temperature reduced a little bit. But it takes much less steps to reach this end phase. In group 2, it takes only 116 steps.

5.3.3 Weld speed 200mm/min, Rotation speed 1750 rpm

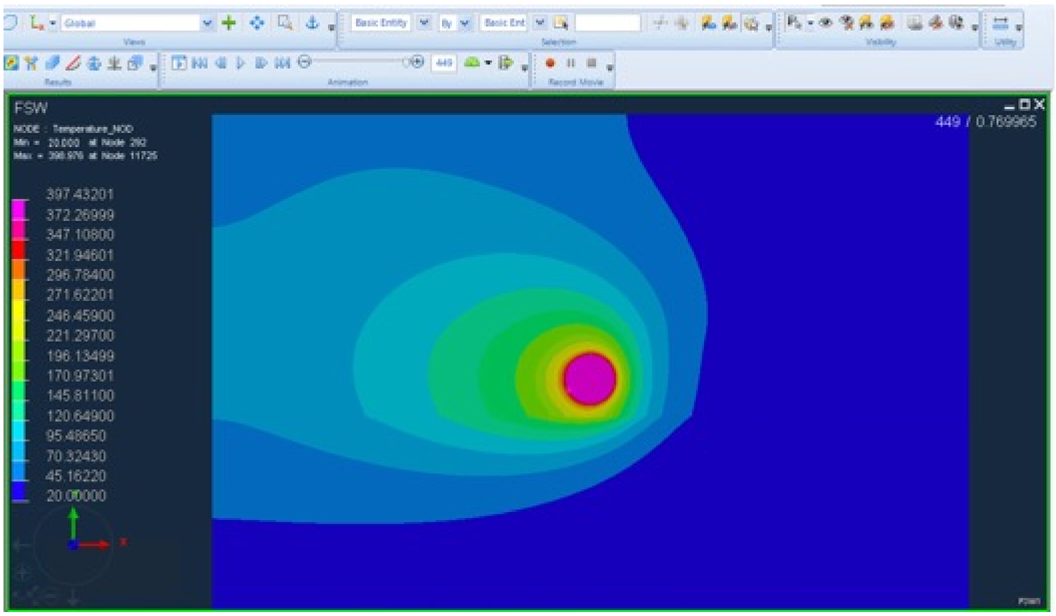


Figure 5.3.3 Temperature field of group 3

From the figure 5.3.3, we can see that the maximum temperature is in the node at tool edge. And the maximum value of temperature is 398.9 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 1 and 2. The maximum temperature is almost the same. But we can see from the contours of the temperature field. Because of the short time to reach its end phase. The temperature is conducted much slower in group 3 than in group 1 and 2. Take the node at middle left point of the sheet, in group 1 it's in the temperature range of 70 degrees.While in group 2 and 3, it's in the temperature range of 45 degrees.

5.3.4 Weld speed 100mm/min, Rotation speed 1250 rpm

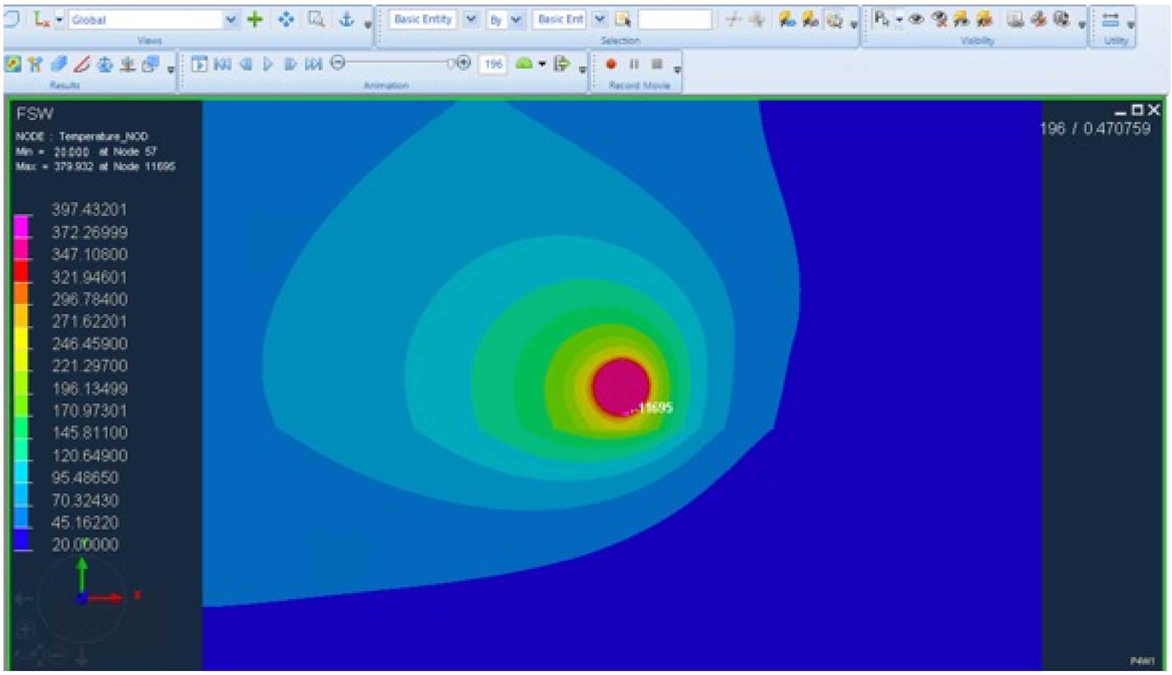


Figure 5.3.4 Temperature field of group 4

From the figure 5.3.4, we can see that the maximum temperature is in the node at tool edge. And the maximum value of temperature is 379.9 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 2, with the same weld speed but group 4 has a slower rotation speed. In this case, group 4 takes a longer time to reach the end phase, which has 196 steps. At the end phase, the maximum temperature is a little bit lower.

But the temperature field distribution is the more or less the same as group 2.

5.3.5 Weld speed 100mm/min, Rotation speed 2250 rpm

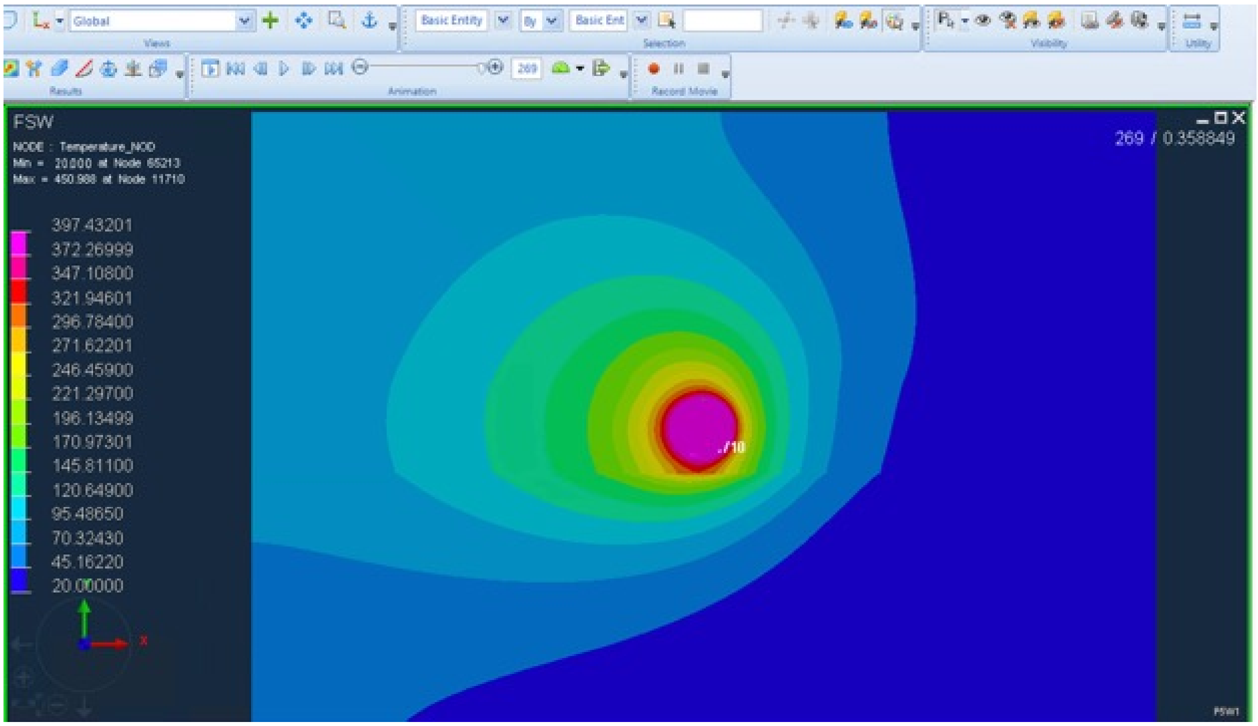


Figure 5.3.5 Temperature field of group 5

From the figure 5.3.5, we can see that the maximum temperature is in the node 11710 at tool edge. And the maximum value of temperature is 450.9 degrees. Compared these 2 different materials, the heat is conducted much faster in the aluminium side than the steel side.

Compared with group 2 and group 4, group 5 has a much higher rotation speed. And it leads to a much higher maximum temperature. In this case, it takes longer time to reach the end phase.

5.4 Conclusion

	Unit	1	2	3	4	5
Tmax in experiment	[C°]	440	367	332	310	368
Tmax in simulation	[C°]	397.4	396.2	398.9	379.9	450.9

Table 5.4.1 Maximum temperature

Comparing the results obtained from the simulation and in the experiment, we can see a difference in the values.

The reason is probably that: in the experiment, the welding sheet has a fixed length along the welding direction, while in the simulation the length is considered as infinite. In this case, in the experiment, the maximum value of the temperature may have not been reached before the welding tool has finished the weldment. And in the simulation result, the process can be relatively long enough for the temperature to reach its maximum.

When the welding speed is increasing, the maximum temperature is decreasing in experiment. Because the duration of the welding process is reduced, so the maximum temperature may have not been reached.

When the rotation speed is increasing, the simulation results show that, the maximum temperature will increase significantly. Because the faster rotation speed increase the heat generation in the contacting face, which leads to a higher temperatures.

5.5 Future developments

By using SYSWELD, I have obtained an amazing result to simulate the friction stir welding process. But during my experience of using this software, I can feel that there need to be more developments for this software to improve the user experience.

From the simulation results, we can view the node temperature and velocity field. But for the further thermo-mechanical results, we would like to have those datas either. During the model building phase, I was always worried about having made any mistakes, and I need to save a copy of current state every few minutes. Because the software does not support the undo command. For the time of setting parameters, it's always hard for me to define the strain rate of a specific material, which is very unrare to look up online and I would like to have some default values of common material for the users to choose.

BIBLIOGRAPHY

[1] M. S. Jaffarullah, “Finite Element Analysis of Stress-Strain Response at the Tool Pin During Friction Stir Process,” Procedia Comput. Sci., p. 6, 2015.

[2] A. B. Verma, S. U. Ghunage, and B. B. Ahuja, “Resistance Welding of Austenitic Stainless Steels (AISI 304 with AISI 316),” p. 6, 2014.

[3] D.A. Price, Williams, A. Wescott, C.J.C Harrison, A. Rezai, A Steuwer, M,Peel, P. Staron and M. Kocak: Sci. Technol. Weld. Join. 2007,12:620-630

[4] Y.D. Chung, H. Fujii, R. u=Ueji and N. Tsuji. Friction stir welding of high carbon sttel with excellent toughness and ductility. Scripta Materialia, 63(2010): 223-226

[5] Hidetoshi Fujii, Ling Cui, Nobuhiro Tsuji, Masatsu Maeda, Kazuhiro Nakata, Kiyoshi Nogi. Friction stir welding of carbon steels. Materials Science and Engineering A, 429(2006):50-57

[6] Ling Cui, Hidetoshi Fujii, Nobuhiro Tsuji and Kiyoshi Nogi. Friction Stir welding of a high carbon sttel. Scripta Materalia 56(2007):637-640

[7] B,C. Liechty, B.W. Webb. The use of plasticine as an analog to explore material flow in friction stir welding [J]. Journal of Materials Processing Technilogy, 2007,(184): 240-250

[8] Saeid T., Abdollah A., Sazgari B. Weldability and mechanical properties of dissimilar aluminum-copper lap joints made by friction stir welding [J]. Journal of Alloys and Compounds, 2010, 490(1-2): 652-655

[9] Sorensen C D, Nelson T W, Packer S M. Tool material testing for FSW of high temperature alloys [C]. 3rd International Symposium on Friction Stir Welding. Kobe, Japan, 2001: 1-7

[10] Nelson T W, Packer S M. Process in polycrystalline cubic boron nitride FSW tooling[C]. 4th International Symposium on Friction Stir Welding. Utah, USA, 2003:1- 14

[11] Sakai Yasushi. Current state and the future of aluminum alloy applications for rolling stock.[J] Journal of Japan Institute of Light Metals. 2006,55(11): 584-587

[12] Threadgill P L, Num M E. A review of friction stir welding: Part 1 process overview[R]. England: TWI, 2003

[13] Liu H J. Friction stir welding assisted by micro-plasma arc[J]. Science and Technology of Welding and Joining, 2009,14(3):193- 195

[14] T.Kasuya, N. Yotiok. Prediction of Welding Thermal History by a Comprehensive Solution [J]. Welding Journal, 1993, 72: 107-115.

[15]U.Dilthey, S.Roosen. Computer Simulation of Thin Sheet Gas-Metal-Arc Welding[J]. Transaction of JWRI, 1996, 25(2): 109-125.

[16]P.Tekri Wal, J.Mazumder. Finite Element Analysis of Three Dimensional Transient Heat Transfer in GMA Welding[J]. Welding Research Supplement, 1998,16:150-156

[17]Lee W B, Lee C Y, Chang W S, et al. Microstructural investigation of friction stir welded pure titanium[J]. Material Letters, 2005, 59(26):3315-3318

[18]Ferrando W A. The Concept of Electrically Assisted Friction Stir Welding and Application to the Processing of Various Metals[R]. Proceedings of the 9th international symposium on friction stir welding. TWI. 2012

[19]Merklein M, Giera A. Laser assisted Friction Stir Welding of drawable steel-aluminum tailored hybrids[J]. International Journal of Mater Form. 2008(1):1299-1302

[20]Ban H S, Song H J, et al. Joint properties of dissimilar Al6061-T6 aluminum alloy/ Ti-6%Al-4%V titanium alloy by gas tungsten arc welding assisted hybrid friction stir welding[J]. Materials and Design, 2013(51): 544-551

[21] Song K H, Tsumura T, Nakata K, Development of microstructure and mechanical properties in Laser-FSW hybrid welded Inconel 600[J]. Materials transactions 2009,50(7):1832